

SEMI-ANNUAL REPORT

Grant No. NAG-849-1

PREDICTION OF FORCES AND MOMENTS
FOR HYPERSONIC FLIGHT VEHICLE
CONTROL EFFECTORS

Mark D. Maughmer
Associate Professor of Aerospace Engineering

Lyle N. Long
Assistant Professor of Aerospace Engineering

Peter J. Pagano
Graduate Research Assistant

Department of Aerospace Engineering
233 Hammond Building
The Pennsylvania State University
University Park, PA 16802
(814) 865-2569

submitted to

National Aeronautics and Space Administration
Langley Research Center
Hampton, Virginia
Attn: David Raney

(NASA-CR-188954) PREDICTION OF FORCES AND
MOMENTS FOR HYPERSONIC FLIGHT VEHICLE
CONTROL EFFECTORS Semiannual Report
(Pennsylvania State Univ.) 48 p CSCL 229

N92-11070

Unclas
63/18 0046464

GRN
12-13-72
4/5/73
P. 4/8

TABLE OF CONTENTS

	<u>PAGE</u>
SUMMARY	2
1. INTRODUCTION	2
Purpose of this work	2
Methodology	3
2. THEORETICAL ASPECTS	4
TEAM Code	4
Surface Inclination Methods	5
Modified Newtonian Theory	5
Tangent Wedge Method	6
Hypersonic Shielding	7
3. TEAM CODE VALIDATION	8
Supersonic/Hypersonic Inviscid Cases	8
Viscous Cases	9
4. PROGRESS	10
Phase I	10
Phase II	10
5. CURRENT AND FUTURE WORK	15
6. REFERENCES	16
APPENDIX: TEAM Code Description	20
FIGURES	25

SUMMARY

The first phase of this research effort, developing methods of predicting flight control forces and moments for hypersonic vehicles, included a preliminary assessment of subsonic/supersonic panel methods and hypersonic local flow inclination methods for such predictions. While these findings clearly indicate the usefulness of such methods for conceptual design activities, deficiencies exist in some areas. Thus, a second phase of research was proposed in which a better understanding is sought for the reasons of the successes and failures of the methods considered, particularly for the cases at hypersonic Mach numbers. This second phase was begun in March, 1990.

To obtain this additional understanding, a more careful study of the results obtained relative to the methods utilized was undertaken. In addition, where appropriate and necessary, a more complete modeling of the flow has been performed using well-proven methods of computational fluid dynamics. As a result of this more complete understanding, assessments will be made which are more quantitative than those of Phase I regarding the uncertainty involved in the prediction of the aerodynamic derivatives. In addition, with improved understanding, it is anticipated that improvements resulting in better accuracy will be made to the simple force and moment prediction methods considered in this study.

1. INTRODUCTION

Purpose of This Work

Included in the executive summary of the AGARD Symposium on the Aerodynamic Characteristics of Controls in 1979 [1] was the need for a more extensive and modern data base. Furthermore, it was suggested that additional research be conducted to fill gaps in the data base. It was also pointed out that theoretical methods were inadequate in accounting for viscous effects and flow separation. More than ten years later, these comments still apply.

In fact, for hypersonic flight vehicles the situation is actually worse. The data base for hypersonic flight control information is extremely limited. Some available wind-tunnel data is of questionable validity and flight-test results are scarce. Furthermore, in addition to the need to account for viscous effects and flow separation, theoretical prediction methods at hypersonic Mach numbers must also contend with problems involving thin shock layers and real gas effects. Some existing computational fluid dynamic (CFD) methods have the ability to handle such problems, but require too much computer and engineering time to be used routinely for conceptual design studies. Consequently, a real need exists for computationally efficient methods of predicting flight control forces and moments for hypersonic vehicles which nevertheless provide reasonable results.

The recent push toward the development of hypersonic flight vehicles has highlighted the need for rapid aerodynamic prediction methods [2 - 4]. The complexity of such vehicles demands the integration of all technological disciplines from the conceptual design stage. Hence, it is of great advantage to be able to analyze many conceptual design proposals and discard those which are not promising in a timely manner. Many methods exist which are capable of performing this analysis; however, at the conceptual design stage monetary and/or time restrictions may preclude their use. CFD techniques are best suited for preliminary or detailed design analysis due to the great length of time required for solution of flows over complex geometries. In addition, the expense and limitations of hypersonic test facilities may relegate their effective use to the testing of final design configurations.

Early integration of control systems into the design process is of paramount importance to the success of a hypersonic vehicle design. At hypersonic Mach numbers, a vehicle traveling through the upper atmosphere will experience dissociation of constituent gases in air. The Space Shuttle Orbiter is a case in point. Upon re-entry, STS-1 required a body flap deflection twice that of the predicted value to trim out the longitudinal moment [5 - 10]. While there is some disagreement over the cause of this problem, most believe it to be due to either real gas effects [11, 12] or low Reynolds number effects [13]. This question is being examined more closely. Other complicating features of hypersonic flows are thick boundary layers, entropy layers, thin shock layers, and boundary layer/shock layer interaction, all of which effect the control aerodynamics. Hypersonic vehicles also experience very large center-of-pressure movements as they traverse the flight envelope from low to high speed. Likewise, the design engineer must consider the changes in flap effectiveness due to the flap being embedded in the boundary layer. It is clear from entropy layer studies [14] that a sharp nosed cone produces a greater pressure recovery on a deflected flap surface than a blunt one. Thus it is concluded that flap effectiveness is decreased by increasing nose bluntness. It is interesting to note that widely used aerodynamic prediction techniques, such as the Aerodynamic Preliminary Analysis System (APAS), other Gentry codes and the Hypersonic Arbitrary Body Program (HABP) make no attempt to model the flow field ahead of a control surface. Flap effectiveness is also decreased as the flap deflection angle is increased. This is caused by the boundary layer separating when the flow is deflected far enough relative to the vehicle body. This separation can create a secondary shock system and/or transition the boundary layer, both of which decrease flap effectiveness. It has also been pointed out that there is a need for a substantial data base [14], which must be shared among the various disciplines involved in hypersonic research.

The purpose of this work is to: 1) to develop and/or improve simple hypersonic aerodynamic methods such as those used in APAS/HABP to better predict viscous and dissociating flow field effects on control surfaces. The initial phase [15] consisted of comparing these simple prediction techniques (data generated using APAS) to experimental data to determine where the greatest need for improvement lies. In this second phase we have been trying to determine which flow phenomena (ie. shockwaves, viscosity, chemistry) have the greatest effect on prediction quality. 2) Add to the database of hypersonic research by examining a relatively simple geometrical shape (the X-15 airfoil) for a range of Mach number, angle of attack, flap deflection and flow field conditions. And 3) attempt to clarify the "hypersonic anomaly" experienced by STS-1.

Methodology

Portions of the three goals of this research project, as outlined in the preceding section, are being attained through the use of an advanced CFD code (the TEAM code), see Chapter 2. This code is essentially being used as a hypersonic wind tunnel. CFD is playing a very important role in the advancement of hypersonic research. Flight testing is a valuable means of collecting data, but it is difficult to accomplish and is often performed post-development. Experimental ground facilities are simply too limited to cover the range of parameters and flight conditions [16]. CFD is not currently the complete solution, there are still many problems to be overcome. It has been suggested, however, that it may provide results as meaningful as those obtained from experimental ground facilities. Such facilities are plagued by the need to extrapolate data to flight conditions, contaminated flow [5], and tunnel peculiar effects on produced data [14]. A major contributor to the tunnel peculiar effects is that of the acoustic environment. The active turbulent boundary layer on the wall of a hypersonic tunnel, as well as any other acoustic disturbances of sufficient strength introduced into the flow-field, will cause transition to occur on the model earlier (at a lower unit Reynolds number) than would be the case for a free-flight experiment [17 - 21].

The lack of experimental hypersonic facilities is yet another impetus for the development and use of CFD codes. A 1968 report to the NASA Subcommittee on Fluid Mechanics of the Committee on Basic Research [22] mentions the need for "wind-tunnel facilities with higher Reynolds number capabilities than are currently available". All the more poignant in the 1990's after decades of inactivity in hypersonic research have depleted the number of operational test facilities. The high cost of hypersonic test facilities has often been their demise. Reference [23] quotes the cost of a Re (based on test section area) = 10 million continuous flow tunnel as over 100 million 1975 dollars.

Thus with an understanding of the shortcomings associated with CFD, and an appreciation of its advantages, a comparison is presented among results from classical Newtonian and tangent wedge theories (both integral components of the APAS/HABP type codes) and results obtained from the TEAM code. Surface pressure plots are compared to illuminate anomalies, and flow-field contour plots from the TEAM code will be shown to explain these differences. Then recommendations will be made for the improvement of the APAS/HABP type analysis methods.

2. THEORETICAL ASPECTS

As discussed in Chapter 1, a comparison is presented between simple hypersonic methods for predicting surface pressure and an advanced CFD technique. The theory pertaining to these various methods is presented here.

TEAM Code

The Three-Dimensional Euler/Navier-Stokes Aerodynamic Method (TEAM) [24] was developed by the Lockheed Aeronautical Systems Company, Burbank, California for the Aeromechanics Division of the Flight Dynamics Laboratory, Wright Research & Development Center under contract of the United States Air Force. The four year (July 1984 - October 1988) effort is the result of a desire to develop a computationally efficient code which could solve both viscous and inviscid flow fields with real-gas effects. Grid system independence was another driving factor in TEAM code development. Grids may be generated using any external program available to the user, only the cartesian coordinates of the grid nodal points are required.

The TEAM code will be briefly describe here, a more detailed description is included in the Appendix of this report. TEAM uses a finite-volume spatial-discretization algorithm coupled to a Runge-Kutta time-marching scheme to solve the Navier-Stokes equations. It can use zonal, patched grids and is therefore quite flexible. It also includes implicit residual smoothing, enthalpy damping, and local time-stepping for efficient convergence to a steady state. The code can simulate turbulent, laminar, or inviscid flow of perfect or real gases.

The TEAM code is normally run on a large supercomputer such as a Cray, Convex, or IBM-3090. As a rough rule of thumb, the code requires 25 microseconds/grid point/iteration. Just to illustrate the CPU requirements of this code, some representative cases are illustrated in the following table. This is just an illustration and individual times for particular runs can vary dramatically depending on the flow field and the grid.

	Inviscid (Euler)		Viscous (Navier-Stokes)	
Dimensions	2-D	3-D	2-D	3-D
No. of Cells	10,000	400,000	20,000	800,000
No. Time Steps	1,000	2,000	10,000	20,000
CPU Time (Cray XMP)	4 Min.	5 Hours	1.4 Hours	111 Hours

This table shows that 3-D, viscous computations are extremely time consuming. It also shows, however, that even a 2-D viscous computation can require hours on a supercomputer. This is one reason for performing the work described herein. In a design environment, one must have tools which are computationally efficient in order to facilitate the iterative nature of design.

Surface Inclination Methods

The nonlinear nature of hypersonic flow manifests itself in such phenomena as high-temperature chemically reacting flow-fields, thin shock layers, entropy layers (vorticity interactions), interactions between the viscous boundary-layer and the shock wave and low-density effects at high altitudes. Considering these phenomena, to obtain a complete picture of the flowfield, one cannot hope to use a simple analytic method. However, there do exist a number of analytic methods, which under certain circumstances provide a good first approximation to the coefficient of pressure (and hence the aerodynamic forces and moments) on a body in a hypersonic inviscid flow-field.

Modified Newtonian Theory

Sir Isaac Newton developed his famous Newtonian flow model more than three centuries ago, it was first published in Propositions 34 and 35 of *Principia* in 1687 [32]. Although developed to explain subsonic flow, this method has seen renewed interest in the latter half of this century as a means of predicting the aerodynamic forces on hypersonic vehicles in the design proposal stage. The equation for this model may be obtained by assuming that a flow impacting on a surface would lose all momentum normal to the surface, and the flow particles would then move tangentially along the surface. Then for a surface inclined at an angle θ to the free-stream,

$$\text{Change in normal velocity} = V_{\infty} \sin \theta$$

$$\text{Mass flux incident on a surface } A = \rho_{\infty} V_{\infty} A \sin \theta$$

$$\text{Time rate of change of momentum of the mass flux} = \rho_{\infty} V_{\infty}^2 A \sin^2 \theta$$

Newton's Second Law states that the time rate of change of momentum is equal to the force exerted on a surface, denoting this force by F ,

$$F = \rho_{\infty} V_{\infty}^2 A \sin^2 \theta \quad (2.12)$$

Newton assumed the flow of particles to be rectilinear, i.e. no random interaction of fluid particles, hence F is associated only with the linear motion of the particles. Static pressure of a gas is due to the purely random motion of its particles, not accounted for in the Newtonian model. Thus F/A ,

which has the dimensions of pressure, must be interpreted as the pressure difference above the free-stream static pressure.

$$\frac{F}{A} = p - p_{\infty} \quad (2.13)$$

Here p is the surface pressure, and p_{∞} is the free-stream static pressure. Combining equations (2.12) and (2.13) and introducing the pressure coefficient gives

$$C_p = 2 \sin^2 \theta \quad (2.14)$$

This result can also be derived from the gas dynamic equations governing oblique shock waves.

Modified Newtonian Theory, as proposed by Lester Lees [33, 34] replaces the coefficient of the sine squared term in equation (2.14) with the coefficient of pressure at the stagnation point behind a normal shock.

$$C_p = C_{p_{\max}} \sin^2 \theta \quad (2.15)$$

Where,

$$C_{p_{\max}} = \frac{P_{s_2} - P_{\infty}}{\frac{1}{2} \rho_{\infty} V_{\infty}^2} \quad (2.16)$$

If the dynamic pressure is written as $\frac{1}{2} \rho_{\infty} V_{\infty}^2 = \frac{\gamma}{2} p_{\infty} M_{\infty}^2$, and use is made of the "Rayleigh Pitot tube formula" [33], the equation for $C_{p_{\max}}$ becomes

$$C_{p_{\max}} = \frac{2}{\gamma M_{\infty}^2} \left(\left[\frac{(\gamma+1)^2 M_{\infty}^2}{4\gamma M_{\infty}^2 - 2(\gamma-1)} \right]^{\frac{\gamma}{(\gamma-1)}} \left[\frac{1-\gamma+2\gamma M_{\infty}^2}{\gamma+1} \right] - 1 \right) \quad (2.17)$$

Notice that in the modified Newtonian theory, C_p is no longer Mach number independent. Further note that as the hypersonic limit is approached (as $M_{\infty} \rightarrow \infty$) and $\gamma \rightarrow 1$, classical Newtonian theory is recovered. Modified Newtonian theory has been shown to be more accurate than straight Newtonian in the prediction of pressures over blunt bodies.

Tangent Wedge Method

The tangent wedge method was developed to predict surface coefficient of pressure on two dimensional hypersonic shapes. Referring to Figure 2.1, suppose it is desired to calculate the pressure at a point i on the body. A line will be drawn tangent to the surface of the body at point i , making an angle θ_i with the free-stream. Then the pressure at point i will be determined as if it were on the surface of a two-dimensional wedge of half-angle θ_i , ie. through the use of exact oblique shock relations. This method assumes that nowhere on the body will the deflection angle

to the free-stream be greater than the maximum turning angle for the free-stream mach number. The tangent wedge method has been shown to work best for sharp nosed bodies with attached leading edge shocks.

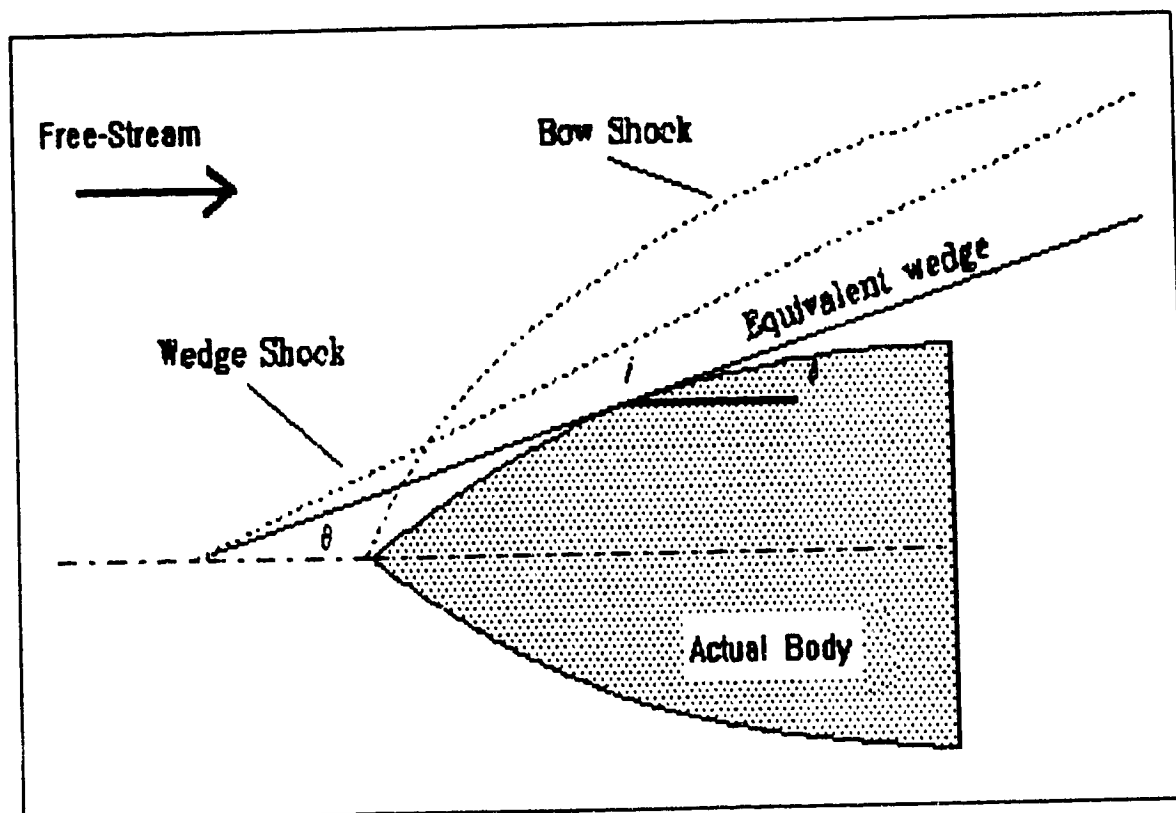


Figure 2.1 Development of the Tangent Wedge Method.

Hypersonic Shielding

Hypersonic shielding is a method used to treat the leeward side of bodies in hypersonic flow-fields. These wake regions are "shielded" from the oncoming free-stream, as a result the surface pressure may be set to zero static pressure. Hypersonic shielding is often used, as it is in this research effort, in conjunction with Newtonian theory, or the tangent wedge method for the impact side of the body.

The definition of pressure coefficient is

$$C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty U_\infty^2}$$

so when $p = 0$ (vacuum) :

$$C_p = \frac{-p_\infty}{\frac{1}{2} \rho_\infty U_\infty^2}$$

and since $\frac{P}{\rho} = RT = \frac{c^2}{\gamma}$, the above becomes :

$$C_p = \frac{-2}{\gamma M_\infty^2}$$

which is the pressure coefficient in a vacuum.

3. TEAM CODE VALIDATION

Experimental verification of the numerical results obtained in this research has not been possible due to an absence of access to a hypersonic wind tunnel facility. Validation of the TEAM code for similar (ie. hypersonic inviscid and viscous) experiments will therefore be cited as verification of the TEAM code.

Supersonic/Hypersonic Inviscid Cases

The first test case to be discussed involves evaluation of TEAM's ability to model attached and detached shocks in supersonic flows. This was accomplished by running TEAM for both a sharp-nose cone cylinder and a blunt-nose cone cylinder at a Mach number of 2.96 and angles of attack of zero and 16 degrees using the standard adaptive dissipation scheme. Additionally, the sharp-nose cone cylinder was modeled at Mach number 4.63 for angles of attack of 4 and 24 degrees. Surface pressure data obtained from TEAM is correlated with experimental data from reference [35].

Good agreement between the experimental and computational results is apparent. Computational and experimental data are again in good agreement except at $\theta = 45^\circ$, here computed surface pressures are below the measured results. The authors of reference [24] cite the absence of viscosity as the most likely cause for the variance.

Evaluation of TEAM's ability to predict hypersonic flow-fields is accomplished by trying to duplicate the experimental results of Shindel [36]. A cone-derived hypersonic waverider was tested at Mach = 6 for angles of attack of -4° , the design angle of attack of 0° , and $+4^\circ$. Computed lift and drag coefficients were in very good agreement with experimental and theoretical values [36]. The data on the upper surface and lower surface out to about 60% correlate well. The disagreement beyond this point on the lower surface is primarily due to the absence of a shock/boundary layer interaction in this region.

A further investigation of hypersonic waverider configurations at off design conditions using the TEAM code has been performed by Long [37]. Inviscid perfect and real gas computations were performed on Rasmussen's elliptical-cone waverider [38]. Computed values of CL, CD and L/D at angle of attack correlated very well with experimental data. Differences between experimental and computational results were attributed to the lack of viscous effects in the computational effort.

Viscous Cases

The first viscous case to be discussed is that of the Lockheed-AFOSR Wing C. This test was conducted at Mach = .85, $\alpha = 5^\circ$ and $R_e = 10$ million based on a MAC of .76 meters. The thin-layer (TLNS) approximation to the RANS was solved in the six zones surrounding the wing, and the Euler equations in the seventh outer zone. Computed surface pressure data are compared with experimental results [39] and an inviscid solution at four stations along the wing. It is evident that the viscous solution better approximates the experimental data. The viscous solution shows a reduction of aft loading and a forward movement of shocks on the wing. It is stated in reference [24] that further research is required to explain the differences between the viscous solution and the experimental data, especially at the outboard stations.

A viscous solution for a double-delta wing-body configuration is another case that was analyzed for the TEAM code validation. This case was run at Mach = .3, $\alpha = 20^\circ$ and $R_e = 1$ million/ft. The TLNS equations were solved in 12 zones next to the body, the Euler equations in the remaining four. Turbulent flow was assumed over the entire surface. Surface pressure data at three cross plane stations for TEAM viscous, TEAM inviscid and experimental are presented in Reference 38. Better over-all prediction at all stations is evident for the viscous solution. Lift and drag coefficients for the viscous case also better approximate the experimental data [24].

Reference [40] is a viscous investigation of an axisymmetric indented nose cone at Mach = 9.89, intended to compare the results from continuum (Navier-Stokes, Euler) methods with those of a kinetic theory approach (Boltzmann equation). The Navier-Stokes method used is TEAM. The kinetic theory approach, as mentioned above, is the DSMC (Direct Simulation Monte Carlo) method developed by Bird [41]. Heat transfer predictions for each method are shown to compare very well to experimental data from reference [42]. Other comparisons of surface pressure coefficient, skin friction coefficient, flow-field density and temperature are also made between the two methods without comparison to experimental data. These flow-field correlations are quite good, and show at least a good agreement between the methods used in TEAM and a solution of the Boltzmann equation.

4. PROGRESS

Phase I

To help address the need for flight control prediction tools, a research program has been underway specifically to provide methods suitable for conceptual design activities involving aerodynamic flight controls. The initial phase of this research included cataloging existing data for hypersonic vehicles and comparing these data with computationally efficient prediction methods. In particular, a preliminary assessment of the subsonic/supersonic panel methods and the hypersonic Newtonian-flow based methods incorporated in the APAS/HABP code [43, 44] has been made. This assessment [45] included a comparison of theoretical predictions with results obtained experimentally for the flight vehicles: the North American X-15, the Hypersonic Research Airplane, and the Space Shuttle. While the experimental data used was taken primarily from wind-tunnel measurements, a few flight-test results for the Shuttle were also included. Comparisons were made from Mach numbers of near zero to twenty.

It was shown that the flow inclination methods do a good job of predicting lift at hypersonic speeds. Most important for flight controls work, the change in lift coefficient due to an elevator deflection is predicted very well. However, the pitching moment versus angle of attack and control deflection angle for the Shuttle at a Mach number of 5.0 is not predicted nearly as well as the lift curve predictions. The change in pitching moment coefficient with control deflection is reasonable, especially at the higher angles of attack. Since separation is not modelled, the results for large control deflections are not good at all. Predicted lateral/directional results (at a Mach number of 5.0) not only agree well with wind-tunnel results, but also agree reasonably well with flight-test data.

This Phase I work has been widely reported. This documentation includes two Master's theses [45, 46], two NASP Contractor Reports [47, 48], and a conference paper [49].

Phase II

During Phase II, the reasons why the impact methods do not accurately predict some aerodynamic coefficients are being explored. The Three-Dimensional Euler/Navier-Stokes Method (TEAM) developed by Lockheed for the US Air Force is being used to accomplish this. This code has been thoroughly tested and can be used as a numerical wind tunnel. It uses a finite-volume, Runge-Kutta algorithm to solve either the Euler equations or the Navier-Stokes equations. It can also model real gas effects and turbulent flow.

To do this, experimental forces and moment data is not sufficient to explain the differences between the impact methods, rather, surface pressure data is required. As experimental data such as this is essentially non-existent for hypersonic Mach numbers, advanced CFD techniques are being used to generate it.

Initially, the X-15 airfoil is being used as a test case. This is a modified NACA 66-005 airfoil. For the purposes of this study, the NACA 66-006 airfoil is scaled down to 5% thickness, since the NACA 66-005 coordinates are not readily available. Unlike the actual X-15 airfoil, the leading edge radius was not scaled and the trailing edge was not blunted. Thus far, at a Mach number of 6.93, angles of attack of 0, 10, and 20 degrees have been considered, with flap deflections of -10, 0, and 10 degrees. At $M=23$, $\alpha = 0$ and 30 degrees with no flap deflections have been considered.

Currently, results for this airfoil have been obtained from both the above described CFD code (in inviscid mode) and from impact theory. Since this is a 2-D problem, instead of using HABP, a simple program developed by Dr. Long is being used. This program, herein called HyperAero, simply uses the modified Newtonian method, the tangent wedge method, and a vacuum condition for surfaces facing away from the free-stream. In the future, APAS/HABP will be used. However, using very simple methods initially allows the determination of what physics must be modelled better in order to more accurately predict the forces and moments. The HyperAero program simply uses the Modified Newtonian method or the Tangent Wedge method for exposed panels and vacuum for hidden panels.

The grid used in the TEAM code is shown in Figure 1 and had 6,144 cells. Several grid sensitivity studies were conducted to determine how fine to make the grid. The grid used is extremely fine, but should result in accuracies within a few percent.

While the above methods are the simplest approximations possible for hypersonic aerodynamics, their limitations are easily quantified. Thus, a great deal can be learned by comparing modern CFD methods to the above methods. In addition, because HyperAero is a Fortran program that is only about 150 lines long, modifications and numerical experiments can be performed very easily. Once it is understood how to obtain accurate predictions for control surface deflections, these can be incorporated into APAS/HABP.

Surface pressure, grey-shaded flow field images, and aerodynamic coefficients are presented below. Figures 2a - 12a show surface pressure predictions (from HyperAero and TEAM) for the Mach = 6.83 inviscid cases. Figures 2b - 12b show the TEAM code flow fields shaded according to pressure for the Mach = 6.83 inviscid cases. The shading actually corresponds to $\ln(p)$, in order to show the gradients better. Since HyperAero predicts only surface quantities (not the whole flow field), the TEAM and HyperAero flow fields cannot be compared directly.

Figures 2a, 3a, and 4a show the surface pressure predictions from TEAM, tangent wedge, and Newtonian flow for $\alpha = 0$ degrees and $\delta = -10, 0$, and $+10$ degrees, respectively, where δ is the flap deflection angle. The Euler code solution is quite smooth, but the HyperAero solution shows a rapid change in C_p at the mid-chord. This is due to the sudden change from the Tangent Wedge method to the vacuum method. In the Tangent Wedge method $C_p = 0$ when $\theta = 0$; however, as soon as $\theta \neq 0$ the method gives the vacuum value for C_p . It is interesting to note that the Euler code produces a smooth result that is approximated by the HyperAero code. It should also be noted that the Euler code predictions are very close to vacuum conditions on the upper surface near the trailing edge, so HyperAero and TEAM agree quite well there. Also, the HyperAero method agrees well with TEAM on the upper surface of the flap since the flow is quite well approximated by a vacuum in that region. The HyperAero predictions on the lower surface of the flap do not agree well with TEAM, however. The tangent wedge method is quite different than TEAM. This severely effects the moment prediction. The tangent wedge method, however, agrees quite well with TEAM near the leading edge.

The impact methods are known to predict low angles of attack poorly. In fact, Hankey [50] claims that more refined methods must be used for angles of attack below 10 degrees. He also says that "only gliders having L/D's greater than 4 will fly at angles of attack less than 10 degrees." So at low angles of attack, one can expect problems in using impact methods.

Figure 5a shows the results of the three prediction methods (TEAM, tangent wedge, and Newtonian) for $\alpha = 10$ and $\delta = 0$ degrees. In this case, the tangent wedge method agrees better with TEAM than the Newtonian method, which is what one would expect. Even though the

tangent wedge method predicts the surface pressure very well over most of the airfoil, however, differences are observed between the TEAM code and tangent wedge at the leading and trailing edges. This immediately indicates a possible error in the moment prediction. The leading edge discrepancy is due to the transition from tangent wedge (or Newtonian) to the vacuum method. The flow transitions from compression to expansion very near the leading edge, but the jump from Newtonian to vacuum is not as noticeable at $\alpha=10$ degrees as it was at $\alpha=0$ degrees. While the TEAM code gradually expands the flow around the corner, the impact methods transition quite abruptly.

In figure 6a the results for $\alpha=10$ degrees and $\delta=10$ degrees are shown. While tangent wedge clearly agrees better with TEAM over most of the airfoil, the Newtonian method once again agrees better over the flap surface. In figure 7a, the results for $\alpha=10$ degrees and $\delta=-10$ degrees are shown. In this case, the flap is aligned with the flow direction, but the airfoil is at an angle of attack. Since the impact methods do not account for any upstream influences, the flap does not "know" the airfoil is at $\alpha=10$ degrees. Consequently, the tangent wedge and Newtonian methods both predict the same C_p 's on the flap that they predict for $\alpha=0$ degrees and $\delta=0$ degrees. The TEAM code flow field, however, has already been turned by the airfoil and must re-expand (lower surface) or re-compress (upper surface) the flow. This discrepancy in the impact methods may cause a significant error in the moment and, possibly, even the lift.

Figure 8a shows the results for $\alpha=20$ degrees and $\delta=0$ degrees. At these higher angles of attack, the impact methods become quite effective and the tangent wedge method clearly agrees better with TEAM. However, you can still see the discrepancies at the leading edge due to the rapid transition from tangent wedge to vacuum method. There are also inconsistencies at the trailing edge.

Figure 9a and 10a show the surface pressure predictions for $\alpha=20$ degrees and $\delta=10$ degrees and $\delta=-10$ degrees, respectively. For $\delta=10$, neither the Newtonian or tangent wedge method agree with TEAM on the flap. This case results in very strong shockwaves, one starting at the leading edge and the other starting at the flap leading edge. These are clearly visible in the flow field shown in Figure 9b. It will be important to obtain viscous results for this case, since significant shock/boundary layer interactions probably exist. The Euler results cannot be assumed correct for this case.

For the negative flap deflection (Figure 10a), the tangent wedge method does a fairly good job predicting the TEAM code results.

Figures 11 and 12 show predictions for $Mach = 23$. At $\alpha = 0$ degrees, the methods all agree fairly well except at the leading edge. At $\alpha=30$ degrees, the shockwave is extremely strong and both Newtonian and tangent wedge deviate from the TEAM code results. It will be important to evaluate the real gas effects at this Mach number.

In all the flow field images, one can clearly see the shock wave starting at the leading edge. For a point source travelling at $Mach = 6.83$, one would expect a Mach cone angle of

$$\beta = \frac{1}{M} = \frac{1}{6.83} = 0.145 \text{ radians}$$

$$\beta = 8.4 \text{ degrees}$$

This is roughly the shock angle near the nose shown in Figure 2b ($\alpha = 0$ degrees). If the airfoil

under consideration were a flat plate, the relation between the shock angle, beta, and the angle of attack, alpha would be governed by

$$\frac{\beta}{\alpha} = \frac{\gamma+1}{4} + \sqrt{\frac{(\gamma+1)^2}{4} + \frac{1}{M^2 \alpha^2}}$$

So for alpha = 10 degrees, Mach = 6.83, and gamma = 1.4, one would get:
 $\beta = 16.3 \text{ degrees}$

So the angle between the airfoil and the shock would be roughly 6.3 degrees. This is roughly what is observed in Figure 5b. The Newtonian flow method assumes the shock wave lies directly on the body surface, which is only true for infinite Mach number and gamma=1.0.

The surface pressure plots and flow field images presented in this section provide an enormous amount of data. From the above discussion one can see how readily the source of the errors in the impact methods can be determined. During Phase I of this grant numerous problems were uncovered in the impact methods, but force and moment data alone is not enough information to propose modifications to the impact methods or their usage.

The force and moment predictions for the preceding cases are summarized in Table 1. While, it is often difficult digesting large tables of numerical data, it is included for completeness. These data illustrate how well the force and moment data can agree, even though the surface pressure may be poorly predicted. They also show how, in other cases, the different prediction methods can disagree by a large margin.

In conclusion, one must be careful in interpreting the results presented here. So far, only inviscid (i.e. Euler equation) TEAM code results have been presented. One cannot assume that these are 100% correct. In some of the cases presented, viscous effects may be quite significant. These will be presented in the next progress report and are being run at this time. It will be important to compare all the methods. While it is quite legitimate to compare inviscid methods to tangent wedge, Newtonian, and vacuum techniques, since they can all be derived from inviscid gas dynamics; methods such as modified Newtonian have some empiricism built into them. Thus, while it will be strictly fortuitous when methods based solely on inviscid techniques agree with viscous results; the empirically based methods may be expected to predict some viscous behavior since they have been "tuned" to yield correct results.

Table 1. Forces and Moment Predictions

Method	Alpha (Deg.)	Delta (Deg.)	CL	CD	CM
Newtonian	0	0	0.0000	0.0092	-0.0000
Tang. Wdg.	0	0	0.0000	0.0114	-0.0000
TEAM	0	0	0.0000	0.0110	-0.0000
Newtonian	0	+10	0.0106	0.0103	-0.0096
Tang. Wdg.	0	+10	0.0229	0.0140	-0.0207
TEAM	0	+10	0.0074	0.0122	-0.0069
Newtonian	0	-10	-0.0106	0.0104	0.0096
Tang. Wdg.	0	-10	-0.0229	0.0140	-0.0207
TEAM	0	-10	-0.0074	0.0121	0.0069
Newtonian	10	0	0.088	0.0248	-0.0346
Tang. Wdg.	10	0	0.160	0.0037	-0.0686
TEAM	10	0	0.121	0.0331	-0.0502
Newtonian	10	+10	0.111	0.0335	-0.057
Tang. Wdg.	10	+10	0.189	0.0507	-0.097
TEAM	10	+10	0.141	0.416	-0.069
Newtonian	10	-10	0.077	0.0236	-0.0251
Tang. Wdg.	10	-10	0.137	0.0353	-0.0480
TEAM	10	-10	0.111	0.0310	-0.0407
Newtonian	20	0	0.231	0.0948	-0.105
Tang. Wdg.	20	0	0.331	0.1312	-0.156
TEAM	20	0	0.315	0.1282	-0.145
Newtonian	20	+10	0.267	0.1198	-0.144
Tang. Wdg.	20	+10	0.371	0.1616	-0.200
TEAM	20	+10	0.356	0.1578	-0.190
Newtonian	20	-10	0.208	0.0863	-0.083
Tang. Wdg.	20	-10	0.303	0.1187	-0.128
TEAM	20	-10	0.292	0.1170	-0.122

5. CURRENT AND FUTURE WORK

Thus far, APAS/HABP, TEAM, and HyperAero for hypersonic flows have been used in this study. The early part of Phase II consisted of setting up the programs and gaining experience in their use. Now that the codes are all in place, with color graphics interfaces, it is expected that the progress will be significant and steady. As described above, investigations using the Euler code results for the study of forces and moments due to control deflections are underway. In the near future, Navier-Stokes (i.e. viscous) and real gas effects will be explored. Unlike experiments or flight tests, CFD allows viscous and real gas effects to be included one at a time or simultaneously.

Also, much of the corroborative development of experimental, analytical, and CFD that took place in the other speed regimes never took place for hypersonic flows because of the hiatus of work in hypersonic flow. There are few detailed comparisons between the simple hypersonic impact methods and CFD -- especially for control deflections.

The space shuttle experienced a severe pitching moment that was not predicted before the first flight. This could have caused a fatal accident if large body flap deflections had not been possible. This problem will be investigated using TEAM, APAS/HABP, and HyperAero to determine why the moments were so poorly predicted. This will also demonstrate how to model the critical physics in APAS/HABP in order to simulate such flows.

6. REFERENCES

1. Young, A. D., Executive Summary, Aerodynamic Characteristics of Controls, AGARD CP-262, May 1979.
2. Burgess, M. A., Schreffler, E. S., Hill, J. C., Osterbeck, P. G., Woodhead, G. E., "HAV-SPRINT: A Conceptual Design Computer Code for Hypersonic Vehicle Synthesis and Optimization," AIAA 87-2956, AIAA/AHS/ASEE Aircraft Design, Systems and Operations Meeting, Sept. 1987.
3. Hunt, J. L., "Hypersonic Airbreathing Vehicle Design (Focus on Aero-Space Plane) Hypersonics," Volume 1: Defining the Hypersonic Environment, Progress in Scientific Computing, Volume 8, Birkhauser, Boston, 1989.
4. Rao, D. M., "Hypersonic Control Effectiveness Studies on Delta Wings With Trailing Edge Flaps," Ph.D. Dissertation, Univ. of London, May, 1970.
5. Bertin, J. J., "General Characterization of Hypersonic Flows," Hypersonics, Volume 1: Defining the Hypersonic Environment, Progress in Scientific Computing, Volume 8, Birkhauser, Boston, 1989.
6. Woods, W. C., Arrington, J. P., and Hamilton, H. H. II, "A Review of Preflight Estimates of Real-Gas Effects on Space Shuttle Aerodynamic Characteristics," Shuttle Performance: Lessons Learned, NASA Conference Publication 2283, Part I, March 1983.
7. Griffith, B. J., Maus, J. R., and Best, J. T., "Explanation of the Hypersonic Longitudinal Stability Problem - Lessons Learned," Shuttle Performance: Lessons Learned, NASA Conference Publication 2283, Part I, March 1983.
8. Kirsten, P. W., Richardson, D. F., and Wilson, C. M., "Predicted and Flight Test Results of Performance, Stability and Control of the Space Shuttle From Reentry to Landing," Shuttle Performance: Lessons Learned, NASA Conference Publication 2283, Part I, March 1983.
9. Findlay, J. T., Kelly, G. M., McConnell, J. G., "Shuttle 'Challenger' Aerodynamic Performance From Flight Data-Comparisons with Predicted Values and 'Columbia' Experience," AIAA 84-0485, 22nd Aerospace Sciences Meeting, January 1984.
10. Romere, P. O., and Young, J. C., "Space Shuttle Entry Longitudinal Aerodynamic Comparisons of Flight 2 with Preflight Predictions," Journal of Spacecraft and Rockets, Vol. 20, No. 6, November-December 1983, pp. 518-523.
11. Maus, J. R., Griffith, B. J., and Szema, K. Y., "Hypersonic Mach Number and Real Gas Effects on Space Shuttle Orbiter Aerodynamics," Journal of Spacecraft and Rockets," Vol 21, No. 2, March-April 1984, pp. 136-141.
12. Bruno, C., "Real Gas Effects," Hypersonics, Volume 1, Defining the Hypersonic Environment, Progress in Scientific Computing, Volume 8, Birkhauser, Boston 1989.
13. Koppenwallner, G., "Low Reynolds Number Influence on Aerodynamic Performance of Hypersonic Lifting Vehicles," AGARD CP-428, April, 1987.
14. Neumann, R. D., "Defining the Aerothermodynamic Methodology," Hypersonics, Volume

- 1: Defining the Hypersonic Environment, Progress in Scientific Computing, Volume 8, Birkhauser, Boston 1989.
15. Ozoroski, L. P., "Evaluation of Methods for Predicting the Forces and Moments Generated by Control Surface Deflections on Hypersonic Vehicles," Master's Thesis, Department of Aerospace Engineering, The Pennsylvania State University, May 1990.
 16. Trella, M., "Introduction to the Hypersonic Phenomena of Hemes," Hypersonics, Volume 1: Defining the Hypersonic Environment, Progress in Scientific Computing Volume 8, Birkhauser, Boston 1989.
 17. Anderson, J. D. Jr., "Hypersonic and High Temperature Gas Dynamics," McGraw-Hill Book Company, New York, 1989.
 18. Morkovin, M. V., "Instability, Transition to Turbulence and Predictability," AGARD-AG-236, Fluid Dynamics 2-4, Panel Symposium on "Laminar-Turbulent Transition," May 2-4, 1977.
 19. Morkovin, M. V., "Critical Evaluation of Transition From Laminar to Turbulent Shear Layers with Emphasis on Hypersonic Traveling Bodies," AFFDL-TR-68-149, March 1969.
 20. Kendall, J. M., "Wind Tunnel Experiments Relating to Supersonic and Hypersonic Boundary-Layer Transition," AIAA Journal, Vol. 13, NO. 3, March 1975, pp. 290-299.
 21. Beckwith, I. E., "Development of a High Reynolds Number Quiet Tunnel for Transition Research," AIAA Journal, Vol. 13, No. 3, March 1975, pp. 300-314.
 22. Howe, J. T., "Some Fluid Mechanical Problems Related to Subsonic and Supersonic Aircraft," NASA SP-183, 1968.
 23. Jones, J. L., "The Transonic Reynolds Number Problem," NASA CP-2009, Workshop on High Reynolds Number Research, October 27-28, 1976, pp. 1-18.
 24. Raj, P., Brennen, J.E., Keen, J.M., Long, L.N., Sikora, J.S., and Singer, S.W., "Three-Dimensional Euler Aerodynamic Method (TEAM)," Vol. 1, 2, and 3, AFWAL-TR-87-3074, WPAFB, Ohio, October 1987.
 25. Vincenti, W. G., and Kruger, C. H., Jr., Introduction to Physical Gas Dynamics, Krieger Press, 1986.
 26. Srinivasan, S., Tannehill, J. C., and Weilmuenster, K. J., "Simplified Curve Fits for the Thermodynamic Properties of Equilibrium Air," NASA Reference Publication 1181, August 1987.
 27. Srihivasan, S. and Tannehill, J. C., "Simplified Curve Fits for the Transport Properties of Equilibrium Air," NASA Contractor Report 178411, December 1987.
 28. Baldwin, B. S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-275, January 1978.
 29. Gnoffo, P. A., McCandless, R. S. and Yee, H. C., "Enhancements to Program LAURA for Computation of Three-Dimensional Hypersonic Flow," AIAA Paper 87-0280, 25th

Aerospace Sciences Meeting, Reno, Nevada, January 12-15, 1987.

30. Gnoffo, P. A., "Application of Program LAURA to Three-Dimensional AOTV Flow Fields," AIAA Paper 86-0565, 24th Aerospace Sciences Meeting, Reno, Nevada, January 9-12, 1984.
31. Tannehill, J. C., Holst, T. L., and Rakish, J. V., "Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock," AIAA Paper 75-154, Pasadena, California, 1975.
32. Newton, Sir I.; *Principia*, Univ. of Calif. Press, 1687.
33. Lees, L., "Hypersonic Flow," Fifth International Aeronautical Conference, Los Angeles, 1955, pp. 241-276. Published by The Institute of Aeronautical Sciences, Through the Sherman M. Fairchild Fund.
34. Anderson, J. D., "Fundamentals of Aerodynamics," McGraw-Hill Book Co., New York, 1984.
35. Landrum, E. J., "Wind-Tunnel Pressure Data at Mach Numbers from 1.6 to 4.63 for a Series of Bodies of Revolution at Angles of Attack from -4° to 60° ," NASA TM X-3558, October 1977.
36. Schindel, L. H., "Design of High Performance Hypersonic Missiles," AIAA Paper 82-0391, 20th Aerospace Sciences Meeting, Orlando, Florida, January 11-14, 1982.
37. Long, L. N., "The Off-Design Performance of Hypersonic Waveriders," AGARD CP-428, April 1987.
38. Rasmussen, M. L., Jischke, M. C., and Daniel, D. C., "Experimental Forces and Moments on Core-Derived Waveriders for $M_\infty = 3$ to 5," Journal of Spacecraft and Rockets, Volume 19, No. 6, December, 1982, pp. 592-598.
39. Keener, E. R., "Pressure-Distribution Measurements on a Transonic Low-Aspect Ratio Wing," NASA TM 86683, September 1985.
40. Long, L. N., "A Comparison of Navier-Stokes and Monte Carlo Methods for Hypersonic Flows," AIAA 88-2730, Thermophysics, Plasmadynamics, and Lasers Conference, June 27-29, 1988.
41. Bird, G. A., *Molecular Gas Dynamics*, Clarendon Press, 1976.
42. Holden, M. S., "Studies of the Heat Transfer and Flow Characteristics of Rough and Smooth Indented Noses - Part I, Steady Flows," AIAA Paper No. 86-0384, 1986.
43. Bonner, E., Clever, W., and Dunn, K., "Aerodynamic Preliminary Analysis System II - Part I, Theory," NASA CR-165627, 1981.
44. Gentry, A. E., "Hypersonic Arbitrary-Body Aerodynamic Computer Program (Mark III Version), Vol. 1 - User's Manual," Report DAC 61552, McDonnell-Douglas Corporation, April 1968.

45. Ozoroski, L., "Evaluation of Methods for Predicting the Forces and Moments Generated by Control Surface Deflection on Hypersonic Vehicles," M.S. Thesis, Department of Aerospace Engineering, Penn State University, May 1990.
46. Ozoroski, T., "An Analysis of Delta Wing Aerodynamic Control Effectiveness in Ground Effect," M.S. Thesis, Department of Aerospace Engineering, Penn State University, May, 1990.
47. Maughmer, M., Ozoroski, L., and Straussfogel, D., "Validation of Methods for Predicting Hypersonic Vehicle Control Effectors," NASP CR-1104, July, 1990.
(see also: Progress Report-- Part I, NASA Grant NAG 1-849, April 1990.)
48. Ozoroski, T.A. and Maughmer, M.D.; "An Analysis of Delta Wing Aerodynamic Control Effectiveness in Ground Effect," NASP CR-1106, July, 1990.
49. Maughmer, M.D., Ozoroski, L., Straussfogel, D., and Long, L.; "Validation of Engineering Methods for Predicting Hypersonic Vehicle Control Forces and Moments," AIAA Paper No. 91-2845, Presented at the AIAA Atmospheric Flight Mechanics Conference, New Orleans, Aug, 1991.
50. Hankey, W.; **Re-Entry Aerodynamics**, AIAA Education Series, Wash., D.C., 1988.

APPENDIX: TEAM Code Description

The Three-Dimensional Euler/Navier-Stokes Aerodynamic Method (TEAM) [24] was developed by the Lockheed Aeronautical Systems Company, Burbank, California for the Aeromechanics Division of the Flight Dynamics Laboratory, Wright Research & Development Center under contract of the United States Air Force. The four year (July 1984 - October 1988) effort is the result of a desire to develop a computationally efficient code which could solve both viscous and inviscid flow fields with real-gas effects. Grid system independence was another driving factor in TEAM code development. Grids may be generated using any external program available to the user, only the cartesian coordinates of the grid nodal points are required. Furthermore, a grid may be subdivided into multiple zones, each zone having its own topology, as well each zone may be specified for solution by a different method, ie. "zone 1" may be solved using the Reynolds averaged Navier-Stokes equations, and "zone 2" by using the Euler equations, etc. Zones are specified and "patched" together with a boundary condition data file which is read by TEAM at execution.

The Reynolds averaged Navier-Stokes equations (RANS) are the widely used mathematical models for the flow of a turbulent gas in thermodynamic equilibrium. These are the equations for conservation of mass, linear momentum and energy which have been time averaged. In integral form,

$$\frac{\partial}{\partial t} \int \bar{w} d\Omega + \int_A \bar{F}^c dA = \frac{\sqrt{\gamma} M_\infty}{Re_\infty} \int_A \bar{F}^v dA \quad (2.1)$$

where \bar{w} is the vector of non-dimensionalized dependant variables

$$\bar{w} = \begin{pmatrix} \rho \\ \rho u_i \\ \rho E \end{pmatrix}$$

\bar{F}^c and \bar{F}^v are the convective and viscous flux vectors given by,

$$\bar{F}^c = \begin{pmatrix} \rho u_n \\ \rho u_i u_n + P n_i \\ \rho H u_n \end{pmatrix} \quad (2.2)$$

$$\bar{F}^v = \begin{pmatrix} 0 \\ \tau_{ij} n_j \\ \tau_{ik} u_k n_j - q_n \end{pmatrix} \quad (2.3)$$

Here ρ is the mass density, u_i are the three cartesian velocity components, E is the total energy, \hat{n} denotes a unit normal vector to the surface, q_∞ is the non-dimensional free-stream speed and Re_∞ is the free-stream Reynolds number based on a characteristic length. A subscript n means the dot product with the vector \mathbf{n} has been taken. Standard summation notation is employed with the subscripts i (or j or k) = 1, 2, and 3, which correspond to the cartesian coordinates X , Y , and Z , respectively. $H = E + \frac{P}{\rho}$ is the total enthalpy, and P is the static pressure. For Newtonian fluids, the viscous stress tensor, τ_{ij} , and heat flux, q_j , are related to the mean flow quantities by.

$$\tau_{ij} = \mu_e \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \lambda_e \frac{\partial u_k}{\partial x_k} \delta_{ij} \quad (2.4)$$

$$q_j = - \left(\frac{\gamma}{(\gamma-1)Pr} \right) k_e \frac{\partial T}{\partial x_j} \quad (2.5)$$

where T is the static temperature, Pr is the free-stream molecular Prandtl number, γ is the ratio of specific heats, μ_e and k_e are the effective dynamic viscosity and thermal conductivity respectively, each is the sum of a molecular and a turbulent part. The effective secondary viscosity, λ_e , is defined by Stokes' hypothesis to be $-2\mu_e/3$. Stokes' hypothesis is most accurate for a monatomic gas, and is a reasonable approximation for incompressible air; however, its use in compressible applications is suspect. The viscous stress tensor is assumed to be symmetric, and all variables are non-dimensional, see section 3.1 [24].

For a large range of flight conditions, air may be assumed to be a perfect gas. This model condition of rigid rotating diatomic molecules may be represented by fixing $\gamma = 1.4$, estimating static pressure from the equation of state,

$$P = (\gamma + 1)\rho \left(E - \frac{1}{2} \mu_j \mu_j + H_e \right) \quad (2.6)$$

Sutherland's law is used to estimate the molecular dynamic coefficient of viscosity.

$$\mu = T^{\frac{2}{3}} \left(\frac{1 + \frac{110.4}{\bar{T}_\infty}}{T + \frac{110.4}{\bar{T}_\infty}} \right) \quad (2.8)$$

Here \bar{T}_∞ is the dimensional free-stream temperature, $T = P/\rho$, and thermal conductivity is estimated as $k = T^{0.71}$.

Equilibrium real gas calculations, which become important for the determination of body surface temperature and density at high mach numbers are an option available to the user of TEAM [17, 25]. These calculations are performed using curve fits developed by Srinivasan et al. [26, 27]. The user must specify free-stream static pressure and density, which are used to estimate free-stream temperature, and free-stream specific enthalpy and specific energy, the ratio of which define γ . Values for local static pressure, temperature, speed of sound, viscosity coefficient and thermal conductivity are then determined from the aforementioned curve fits using the estimated values of local specific energy and density.

For viscous computations, the turbulence model used in TEAM is the well known Baldwin-Lomax Turbulence Model (BLTM) [28], or one can use laminar flow. Inviscid computations are carried out by solving the Euler equations. These will not be written explicitly here as they are derived by simply setting the right hand side of equation (2.1) equal to zero. The Euler equations may be solved with either perfect or real gas computation models as previously described.

Numerical dissipation needs to be included in the TEAM code primarily for two reasons, 1) stability of the solution process and 2) shock capturing [24]. Stability must be numerically enhanced, as inviscid or high Reynolds number calculations have little or no physical dissipative phenomena. Without this artificial dissipation, the solution may become "saw-toothed" with alternating signs at neighboring cells. Often solutions of the RANS equations also require numerical

dissipation when the physical (viscous) dissipation is not adequate.

Shock capturing is performed automatically when solving the RANS equations. Euler solutions, on the other hand, do not contain the means to enforce an entropy condition as required by the second law of thermodynamics, hence providing a solution which is not physically realistic. The addition of dissipative terms which imitate the physics inside a shock wave circumvents this error. TEAM provides a choice of three adaptive and two characteristic based dissipation schemes. As this is a study in hypersonics, a characteristic-based scheme is employed. An upwind second order (USO) accurate scheme known as the symmetric TVD formulation [29, 30] allows for the capturing of strong shocks encountered at hypersonic speeds. The price of this improved shock capturing is an increase in the number of arithmetic operations, as compared to adaptive schemes. It is also not possible to satisfy the condition of constant total enthalpy for a steady state Euler solution, which is precipitated by the inconsistency of the steady-state mass and energy conservation equations each of whose dissipative terms are constructed independently.

TEAM requires the user to specify a boundary condition at all grid edges and interfaces. This specification then allows the code to create "ghost cells" beyond the grid boundaries and assign a value of the dependant variable in the image cell which, when the fluxes of the cells (boundary and image) are averaged, gives the proper boundary condition dependant evaluation of the flux vector at the cell face.

Far-field boundary conditions are specified at boundaries where the flow is incoming or outgoing. Hypersonic/supersonic flow dictates that all of the flow quantities in the image cells at inflow boundaries be set to their free-stream values. At the outflow boundary, all image cell quantities are set to their boundary cell values. These criteria are determined by the direction of the characteristics at the corresponding boundaries.

Solid surface boundary conditions are prescribed differently for the Euler equations as compared to the Navier-Stokes equations. Inviscid flow requires the no-normal-flow condition be satisfied at solid boundaries. TEAM provides a choice of three methods to satisfy this condition. That used for this research is the simplest and most robust of the three. Surface pressure is set equal to the cell-center value, this pressure on the cell face is the only variable to contribute to the momentum flux balance. The convective flux may be set to zero at the cell face to preserve the no-normal-flow condition. A surface boundary condition for solution of the RANS equations is the no-slip condition. This condition is imposed by setting the image-cell values of the cartesian components of momentum to be negative of the boundary-cell values, thus insuring the momentum be zero at the surface. The same method of estimating surface pressure as in the inviscid cases was employed for viscous runs. Surface temperature may be prescribed, or an adiabatic condition imposed. In the case of the latter, a zero normal temperature gradient is imposed on the surface to estimate the value of the image-cell temperature.

Grid branch cuts must be specified as "fluid" conditions in the boundary condition dataset. Values for the image-cells on one side of the branch cut are set to those of the boundary-cells across the branch cut, for both sides of the branch cut.

Boundary conditions for planes of symmetry are specified by mirroring the flow field across the plane, eg. across an X-Z plane of symmetry, the Y-component of momentum changes sign while all other variables remain the same as their boundary-cell counterparts. For the 2-dimensional cases used in this research effort, two planes of symmetry were specified with one cell between them.

The semi-discrete approximation which is to be integrated in time is as follows.

$$\frac{d}{dt}(\Omega w) + Q^c - Q^v - D = 0 \quad (2.9)$$

Here Q^c is the convective flux, Q^v is the viscous flux, D is the dissipation and Ω represents the volume. Since the volume Ω is independent of time, equation (2.9) may be rewritten as,

$$\frac{dw}{dt} + R(w) = 0 \quad (2.10)$$

where R is the residual defined as,

$$R(w) = \frac{1}{\Omega}(Q^c(w) - Q^v(w) - D(w)) \quad (2.11)$$

Thus is defined a system of ordinary differential equations which may be solved by a variety of time marching schemes. Time accuracy is not important here as a computationally efficient steady state solution is the goal. TEAM uses an explicit multistage time-stepping scheme. This scheme allows relatively large time-steps and is easily vectorizable to exploit the capabilities of modern supercomputers. This m-stage hybrid scheme can be represented as follows.

$$\begin{aligned} w^{(0)} &= w^0 \\ w^{(1)} &= w^{(0)} - \alpha_1 \Delta t^* R^{(0)} \\ w^{(2)} &= w^{(0)} - \alpha_2 \Delta t^* R^{(1)} \\ &\vdots \\ w^{(m-1)} &= w^{(0)} - \alpha_{m-1} \Delta t^* R^{(m-2)} \\ w^{(m)} &= w^{(0)} - \alpha_m \Delta t^* R^{(m-1)} \\ w^{n+1} &= w^{(m)} \end{aligned}$$

Where $\Delta t^* = CFL \cdot \Delta t$, the Courant number, CFL is a user specified parameter which scales the time step Δt . A pseudo time stepping, or spatially varying time step substantially reduces the number of time steps to convergence. This involves using a local time step for each cell, rather than a globally minimum time step. One consequence of pseudo time stepping is that the solution is no longer time accurate, ie. the solution is meaningless until convergence is reached. Viscous computations often require a much smaller time step than inviscid calculations because they require finer, highly clustered grids. TEAM allows the user to choose between three options for selecting the time step. 1) An inviscid time step in conjunction with two evaluations of numerical and viscous dissipation, 2) application of the modified Crocco's scheme to scale the inviscid time step to satisfy a viscous stability limit, 3) use of a formulation proposed by Tannehill et al. [31], which estimates the time step to automatically satisfy the viscous stability criteria. In this research project, method 1) for inviscid calculations, and method 3) for viscous calculations has been used.

Aerodynamic forces and moments on a body are determined by integrating the normal and tangential stresses. Shear stresses are, of course, absent for an inviscid computation. Denoting normal forces by superscript N , and shear forces by superscript S , force vectors and force coefficients in the body-fixed coordinate system are as follows.

$$\vec{F} = \vec{F}^N + \vec{F}^S \text{ and } \vec{C}_F = \vec{C}_F^N + \vec{C}_F^S$$

$$\vec{F}^N = \int_{A_r} p \hat{n} dA \text{ and } \vec{C}_F^N = -\frac{2}{A_R A_r} \int_{A_r} C_p \hat{n} dA$$

$$\vec{F}^S = -\int_{A_r} \vec{\tau} dA \text{ and } \vec{C}_F^S = -\frac{2}{\gamma M^2 A_R A_r} \int_{A_r} \vec{\tau} dA$$

Here p is static pressure, and $\vec{\tau}$ is the dot product of the stress tensor and the unit normal vector \hat{n} .

The moments and moment coefficients about a point with position vector \vec{r} are given by:

$$\vec{M} = \vec{M}^N + \vec{M}^S \text{ and } \vec{C}_M = \vec{C}_M^N + \vec{C}_M^S$$

$$\vec{M}^N = -\int_{A_r} p (\vec{r} \times \hat{n}) dA \text{ and } \vec{C}_M^N = -\frac{2}{A_R A_r} \int_{A_r} C_p (\vec{r} \times \hat{n}) dA$$

$$\vec{M}^S = -\int_{A_r} \vec{r} \times \vec{\tau} dA \text{ and } \vec{C}_M^S = -\frac{2}{\gamma M^2 A_R A_r} \int_{A_r} \vec{r} \times \vec{\tau} dA \quad (32)$$

Here A_R is the reference area, and c_R denotes the reference chord used in defining the force and moment coefficients. Coefficients of lift, drag and side-force, and the coefficients of the pitching, rolling and yawing moments are obtained in the wind-axis frame through the use of a transformation matrix.

$$\begin{bmatrix} C_L \\ C_D \\ C_Y \end{bmatrix} = \begin{bmatrix} -\sin\alpha & 0 & \cos\alpha \\ \cos\beta \cos\alpha & \sin\beta & \cos\beta \sin\alpha \\ -\sin\beta \cos\alpha & \cos\beta & -\sin\beta \sin\alpha \end{bmatrix} \begin{bmatrix} C_{Fx} \\ C_{Fy} \\ C_{Fz} \end{bmatrix}$$

$$\begin{bmatrix} C_M \\ C_R \\ C_N \end{bmatrix} = \begin{bmatrix} -\sin\beta \cos\alpha & \cos\beta & -\sin\beta \sin\alpha \\ \cos\beta \cos\alpha & \sin\beta & \cos\beta \sin\alpha \\ -\sin\alpha & 0 & \cos\alpha \end{bmatrix} \begin{bmatrix} C_{Mx} \\ C_{My} \\ C_{Mz} \end{bmatrix}$$

The expressions for aerodynamic parameters are for steady flow only.

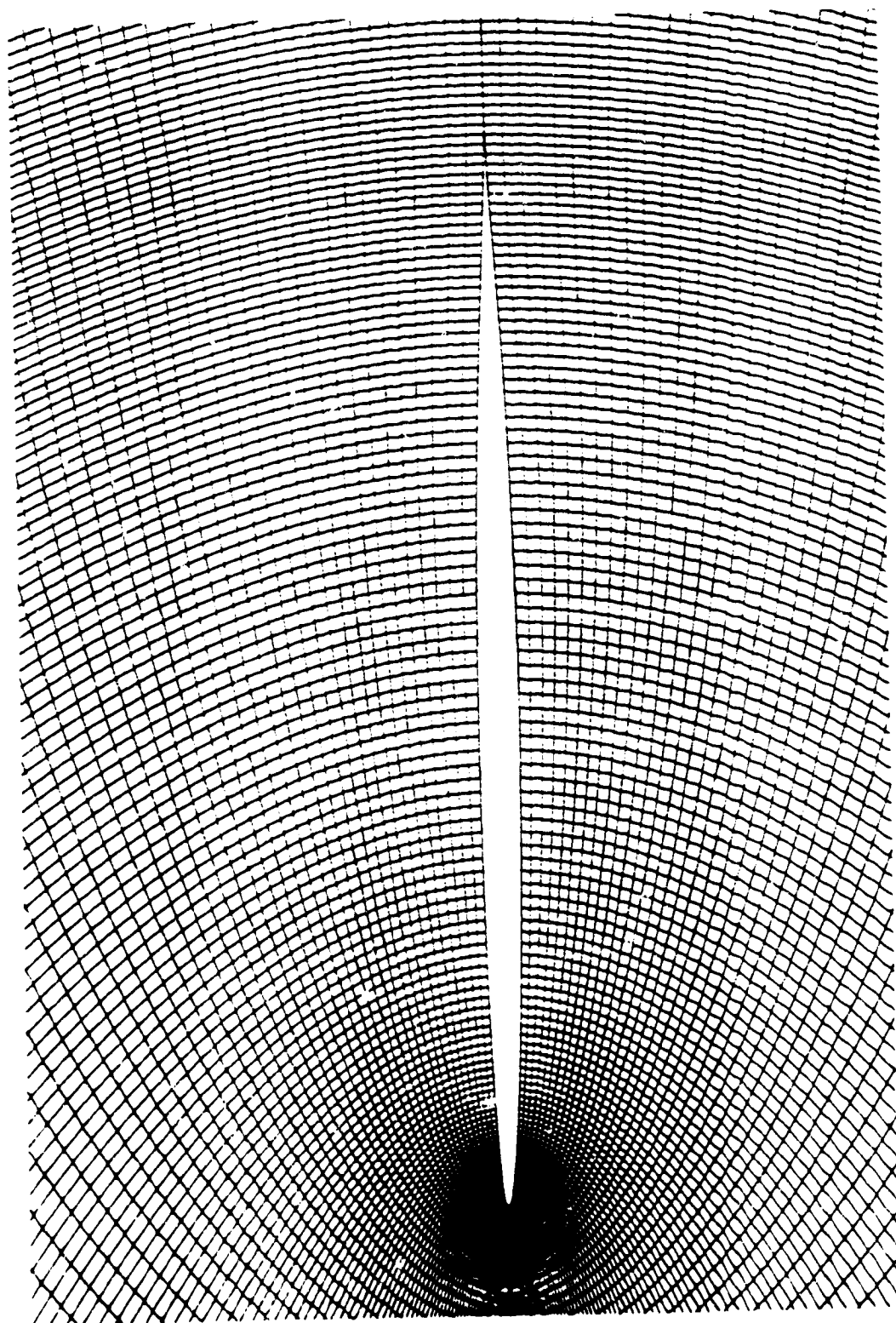
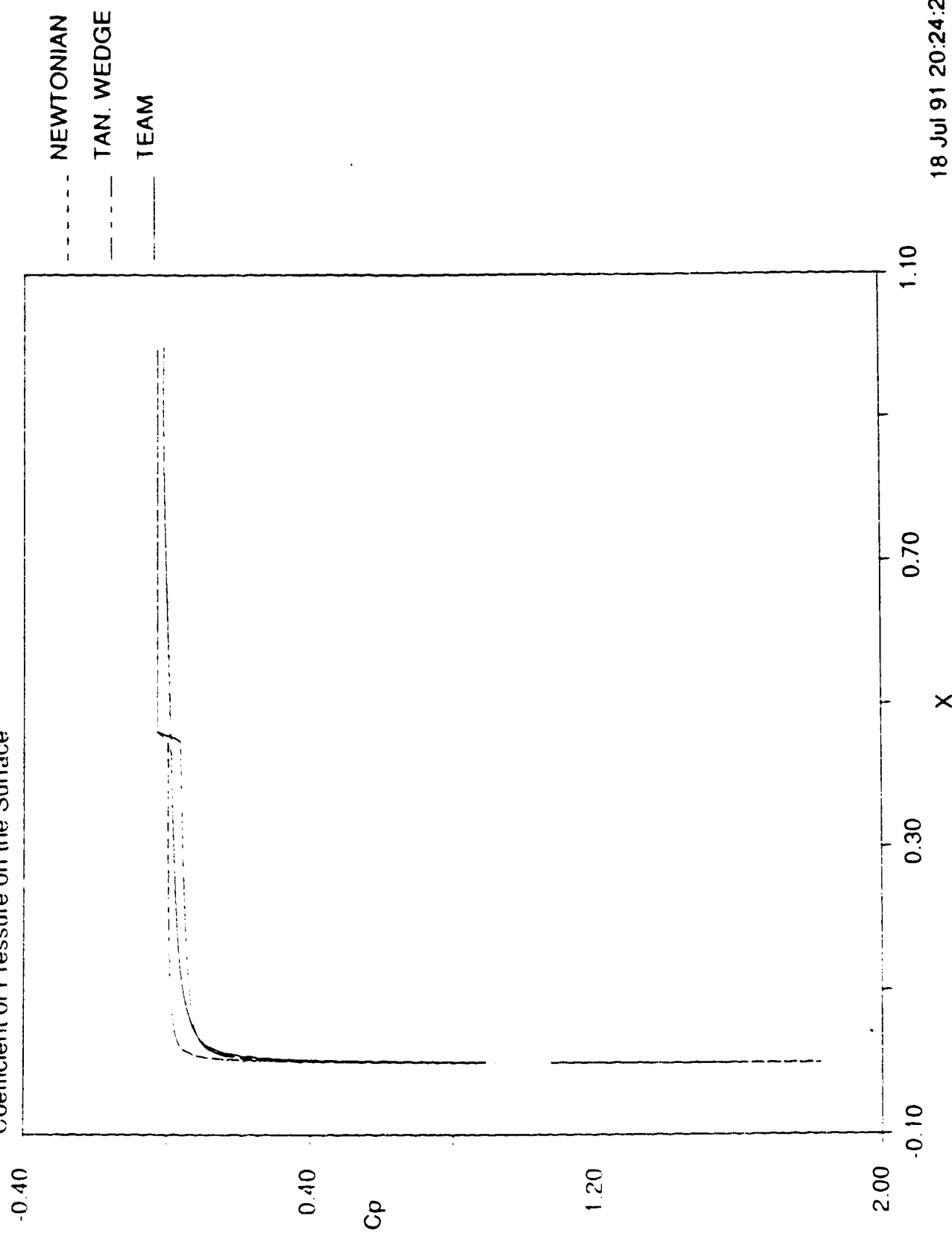


Figure 1 Fine Inviscid Grid

PSU, Hyperaero Results: $M = 6.83$, $A = 0$, $D = 0$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



18 Jul 91 20:24:23

Figure 2a.

PSU, TEAM flow field results: $M = 6.83$, $A = +0$, $D = +0$, Inviscid, Perf. Gas

Natural Log of Static Pressure Ratio

0.65

0.22

Z

-0.22

-0.65
-0.10

1.20

0.77

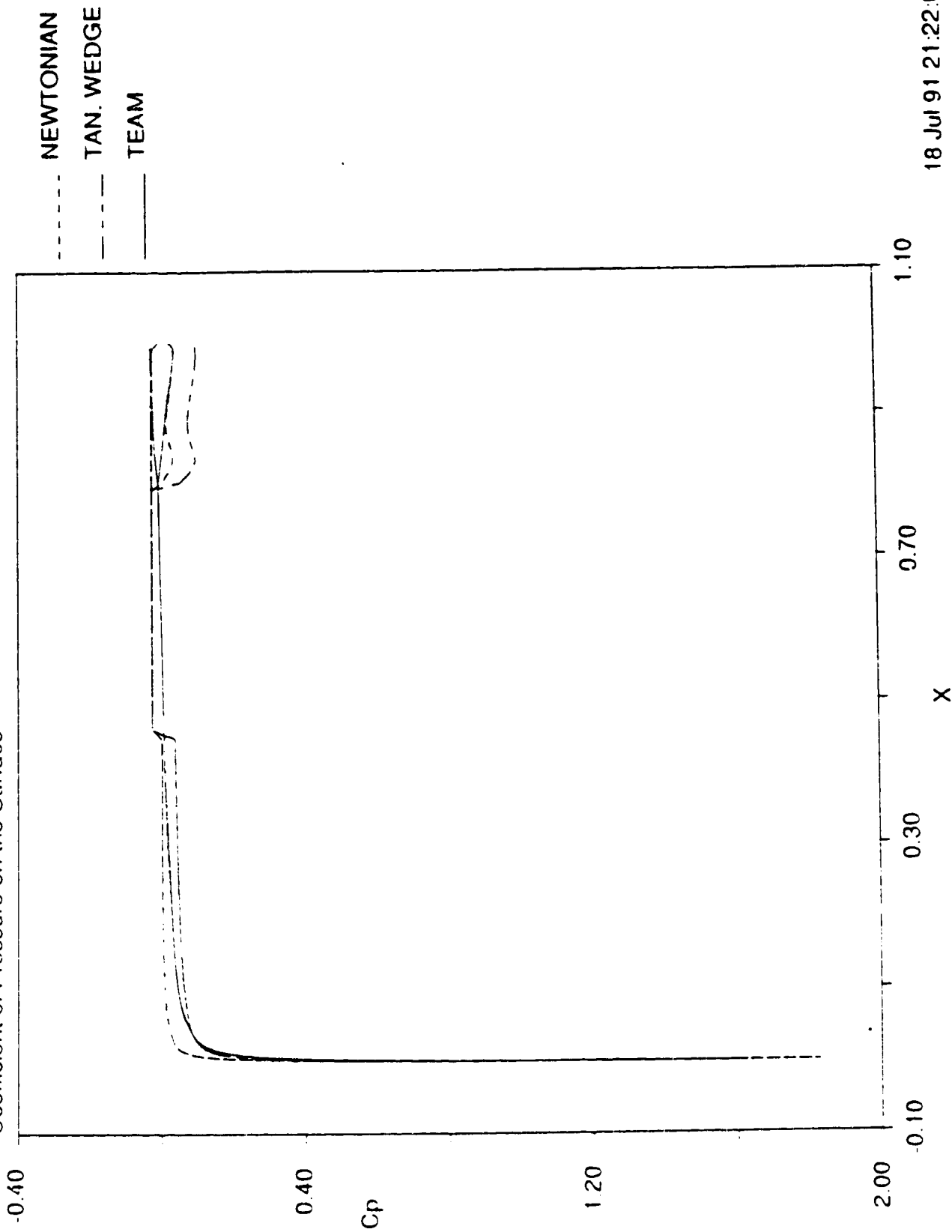
0.33

X

29 Jun 91 15:04:57

Figure 2b.

PSU, Hyperaero Results: $M = 6.83$, $A = 0$, $D = 10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



18 Jul 91 21:22:06

Figure 3a.

PSU, TEAM flow field results: $M = 6.83$, $A = 10$, $D = 10$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

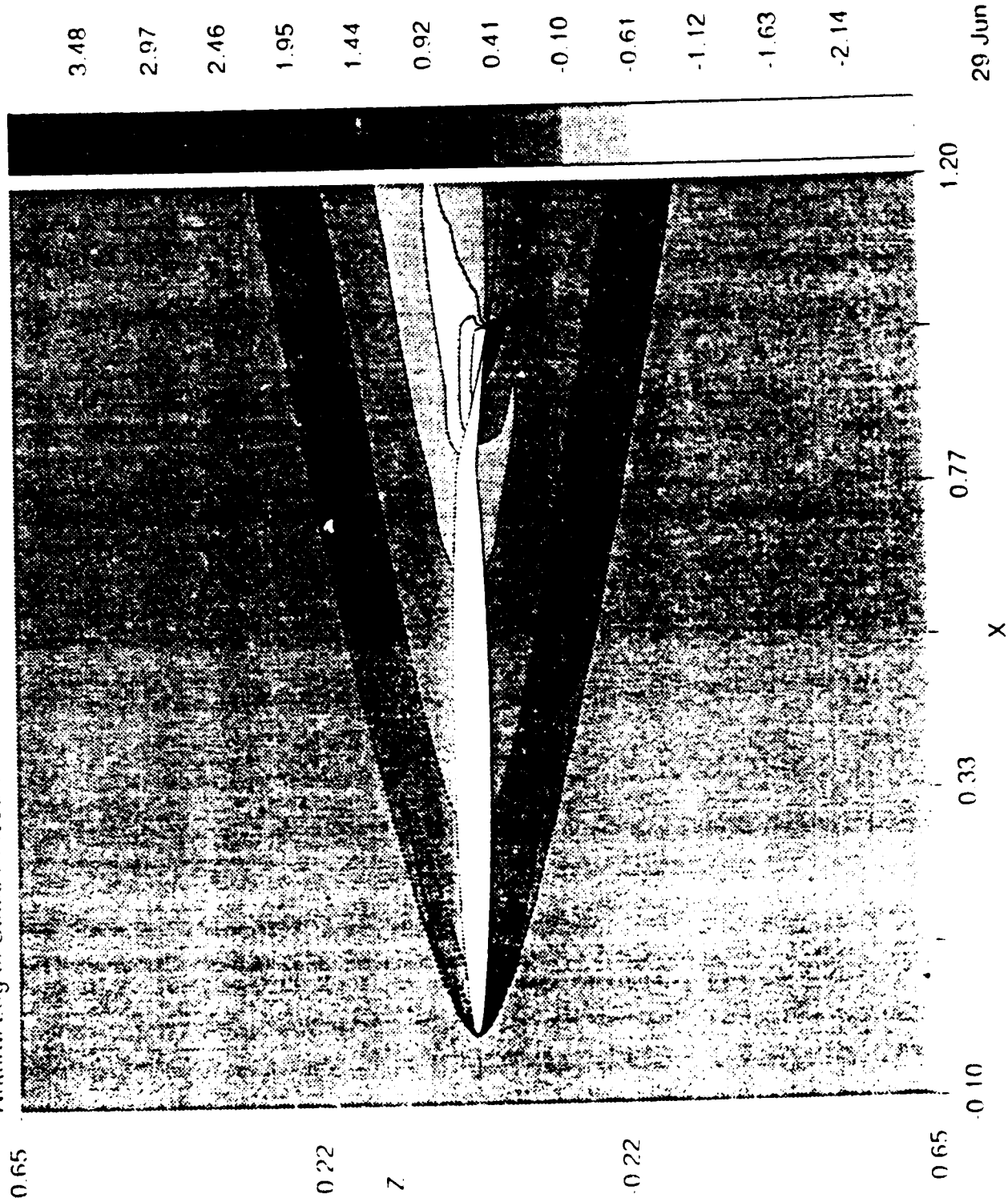


Figure 3b.

29 Jun 91 15:12:46

PSU, Hyperaero Results: $M = 6.83$, $A = 0$, $D = -10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface

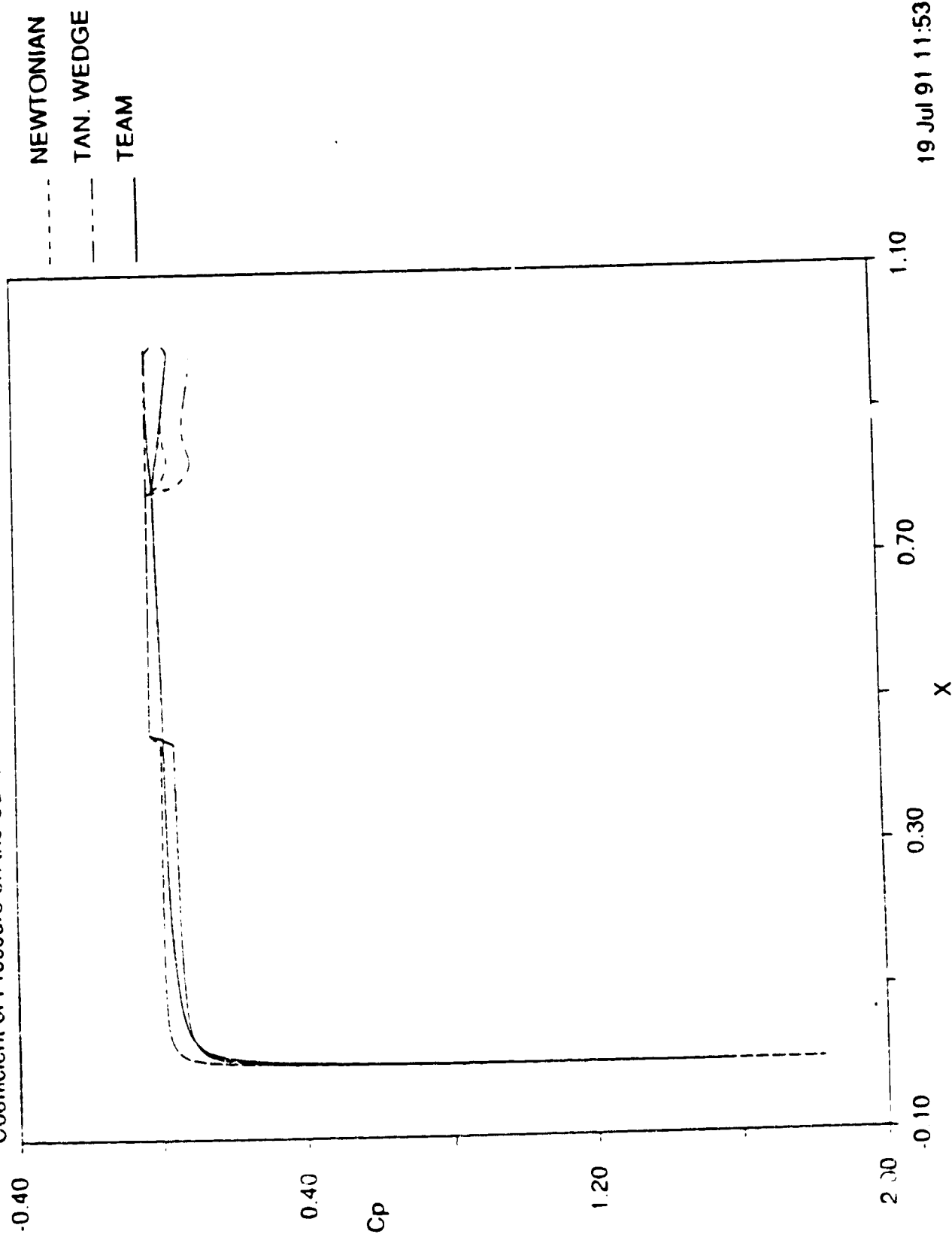


Figure 4a.

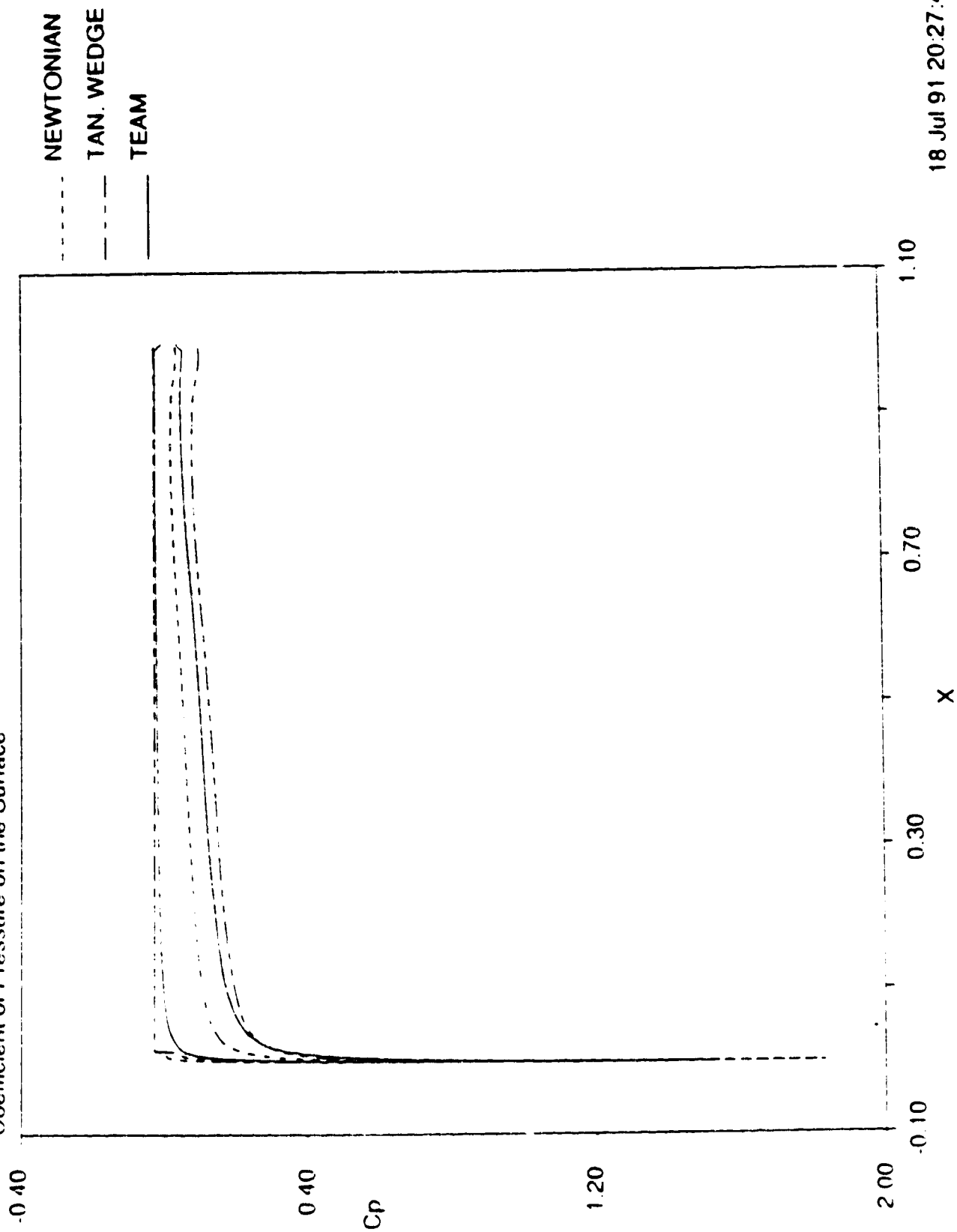
19 Jul 91 11:53:28

PSU, TEAM flow field results: $M = 6.83$, $\Lambda = 10$, $D = 10$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio



Figure 4b.

PSU, Hyperaero Results: $M = 6.83$, $A = 10$, $D = 0$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



18 Jul 91 20:27:42

Figure 5a.

PSU, TEAM flow field results: $M = 6.83$, $A = 10$, $D = 10$, Inviscid, Perf Gas
 Natural Log of Static Pressure Ratio

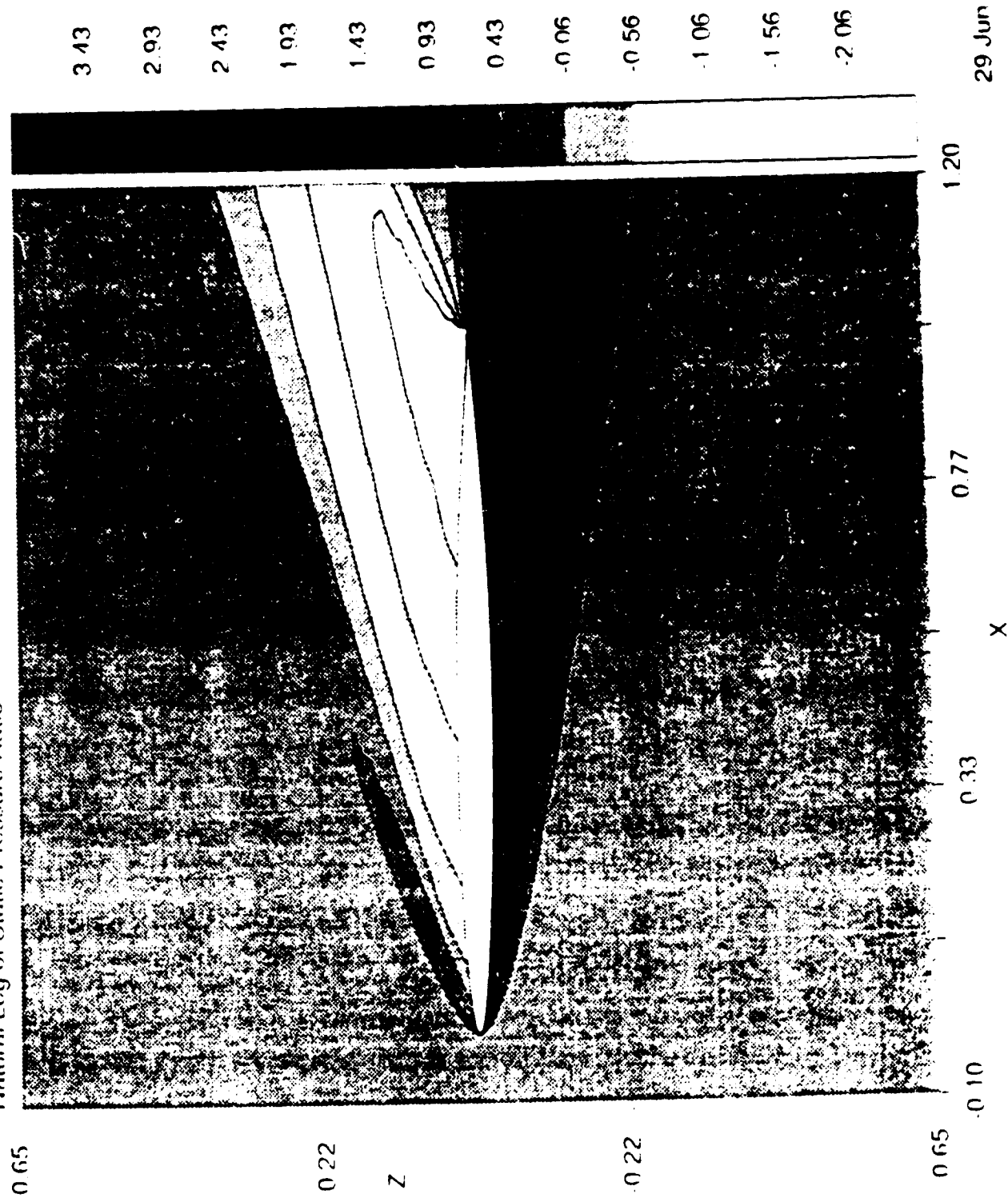


Figure 5b.

29 Jun 91 15:30:26

PSU, Hypersonic Results: $M = 6.83$, $A = 10$, $D = 10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface

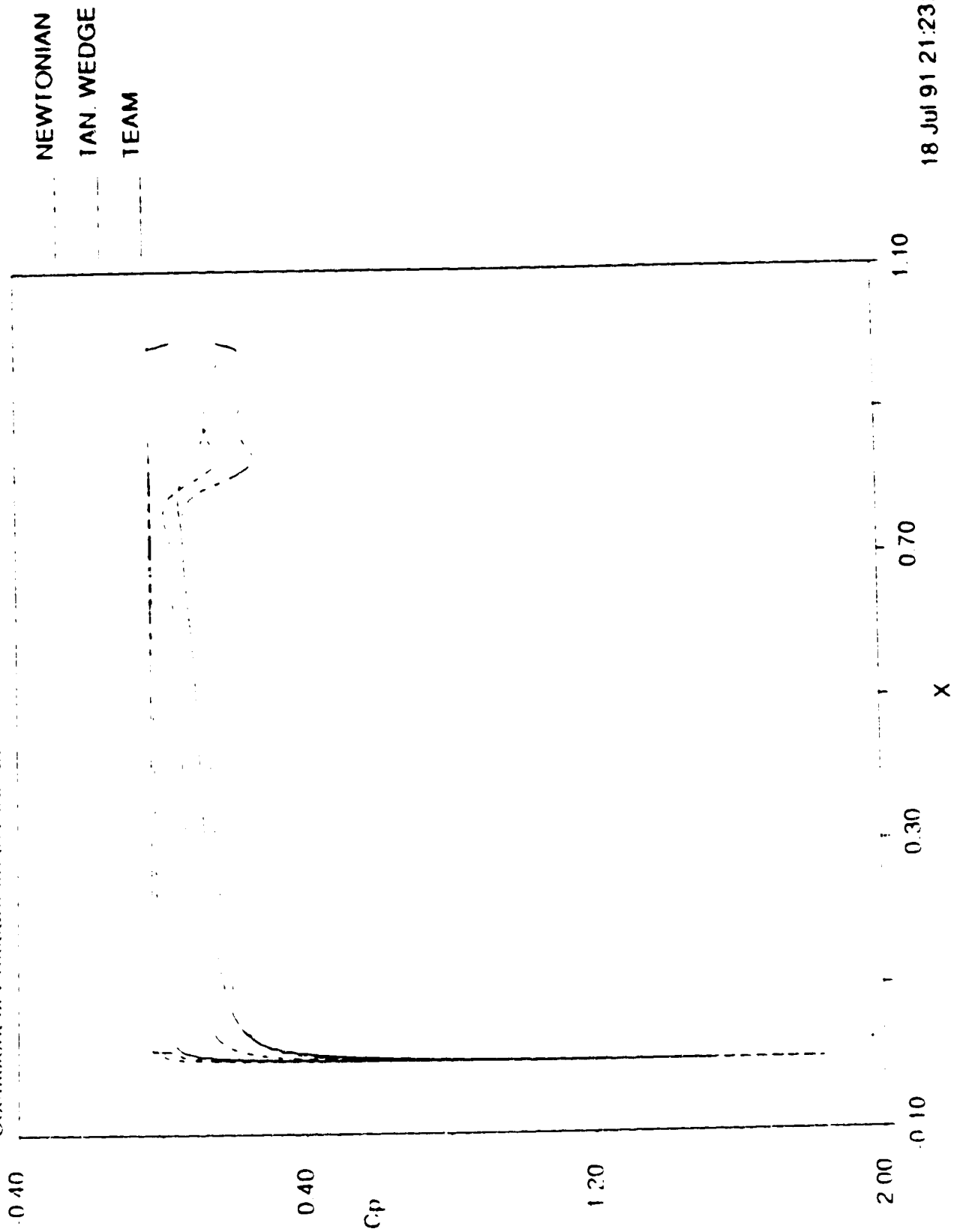


Figure 6a.

18 Jul 91 21:23:50

PSU, IFAM flow field results: $M = 6.83$, $A = +10$, $D = +10$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

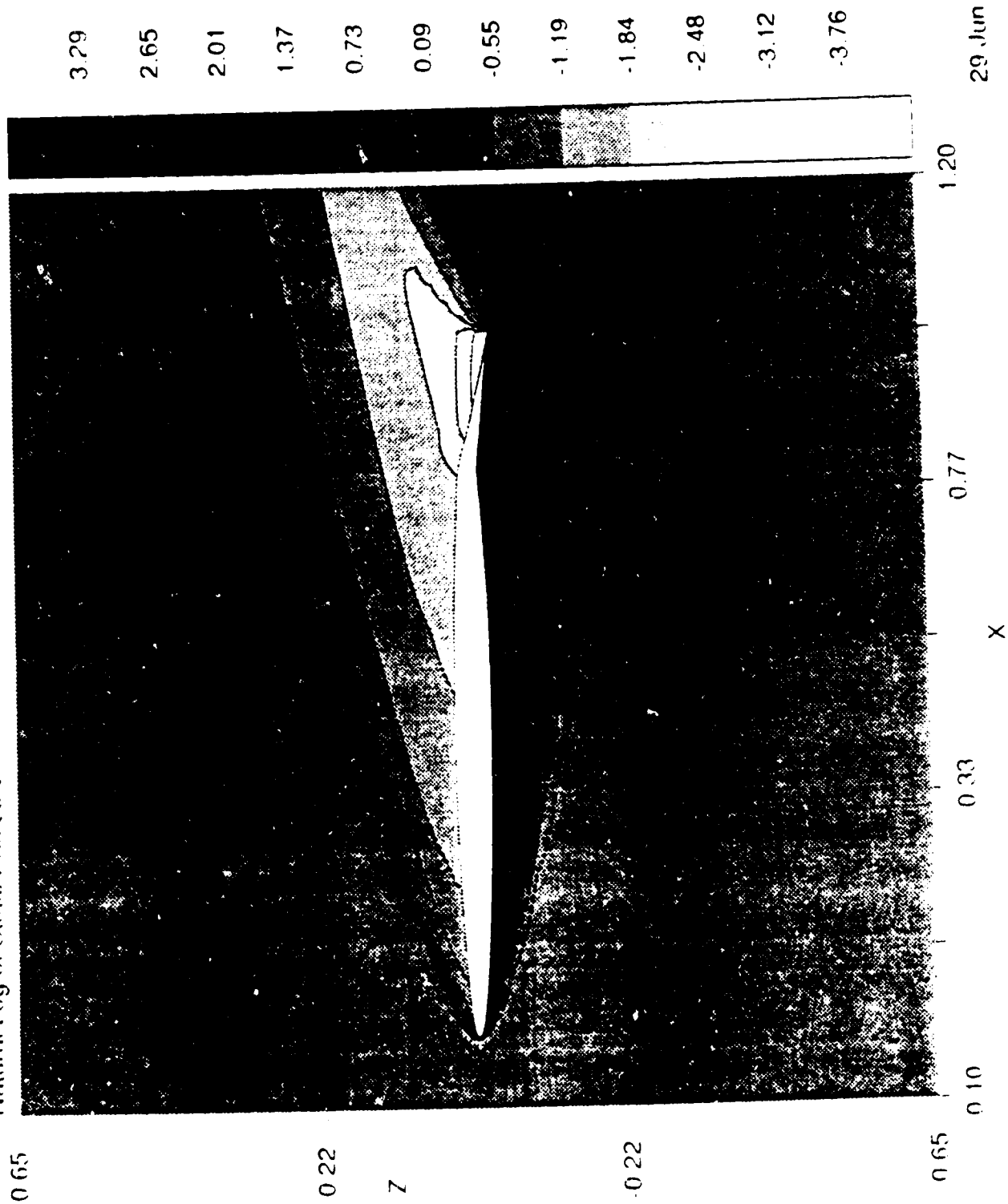


Figure 6b.

29 Jun 91 15:40:37

PSU, Hyperaero Results: $M = 6.83$, $A = 10$, $D = -10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface

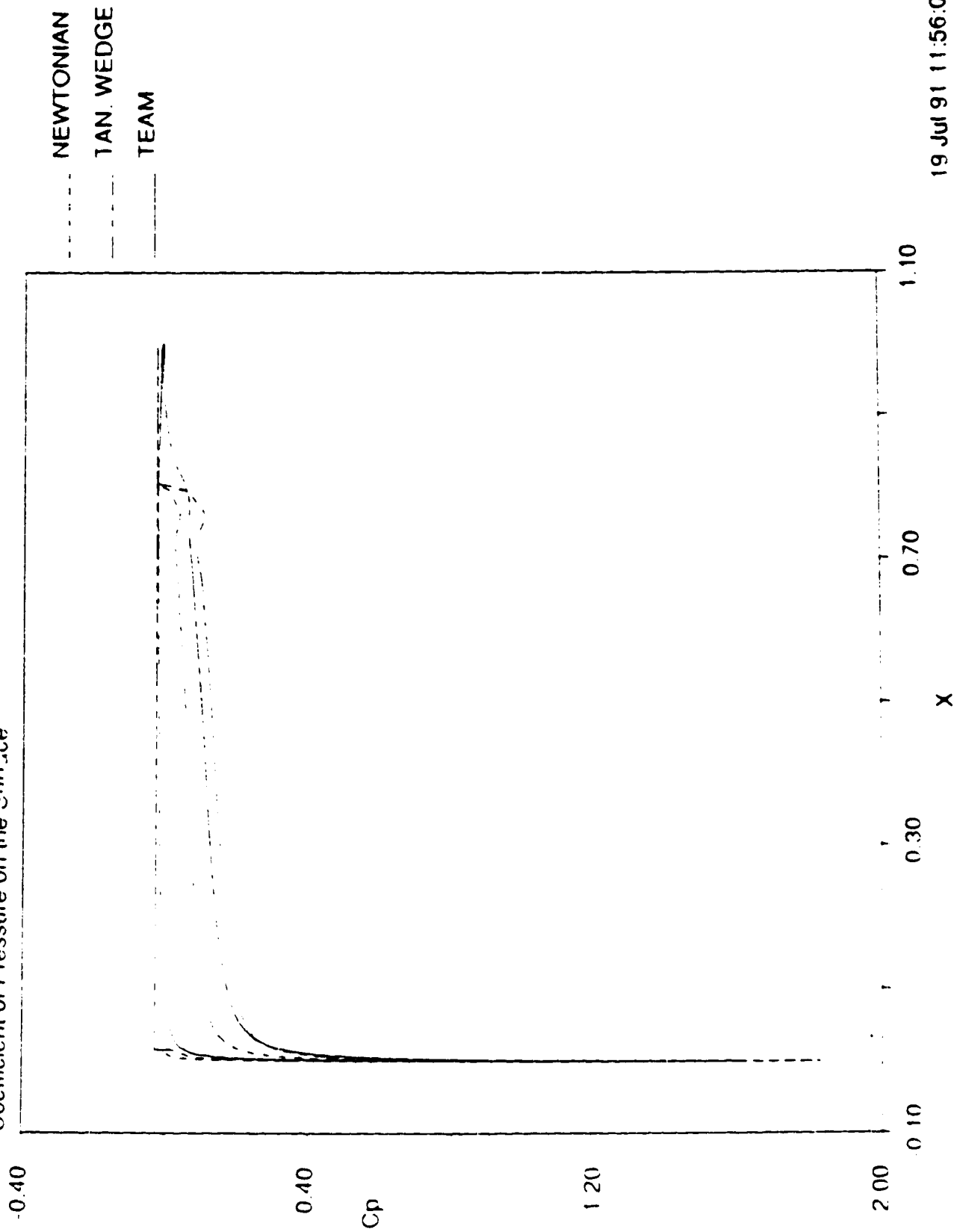


Figure 7a.

19 Jul 91 11:56:01

PSU, IFAM flow field results. $M = 6.83$, $A = 10$, $D = 10$, Inviscid, Perf Gas
 Natural Log of Static Pressure Ratio

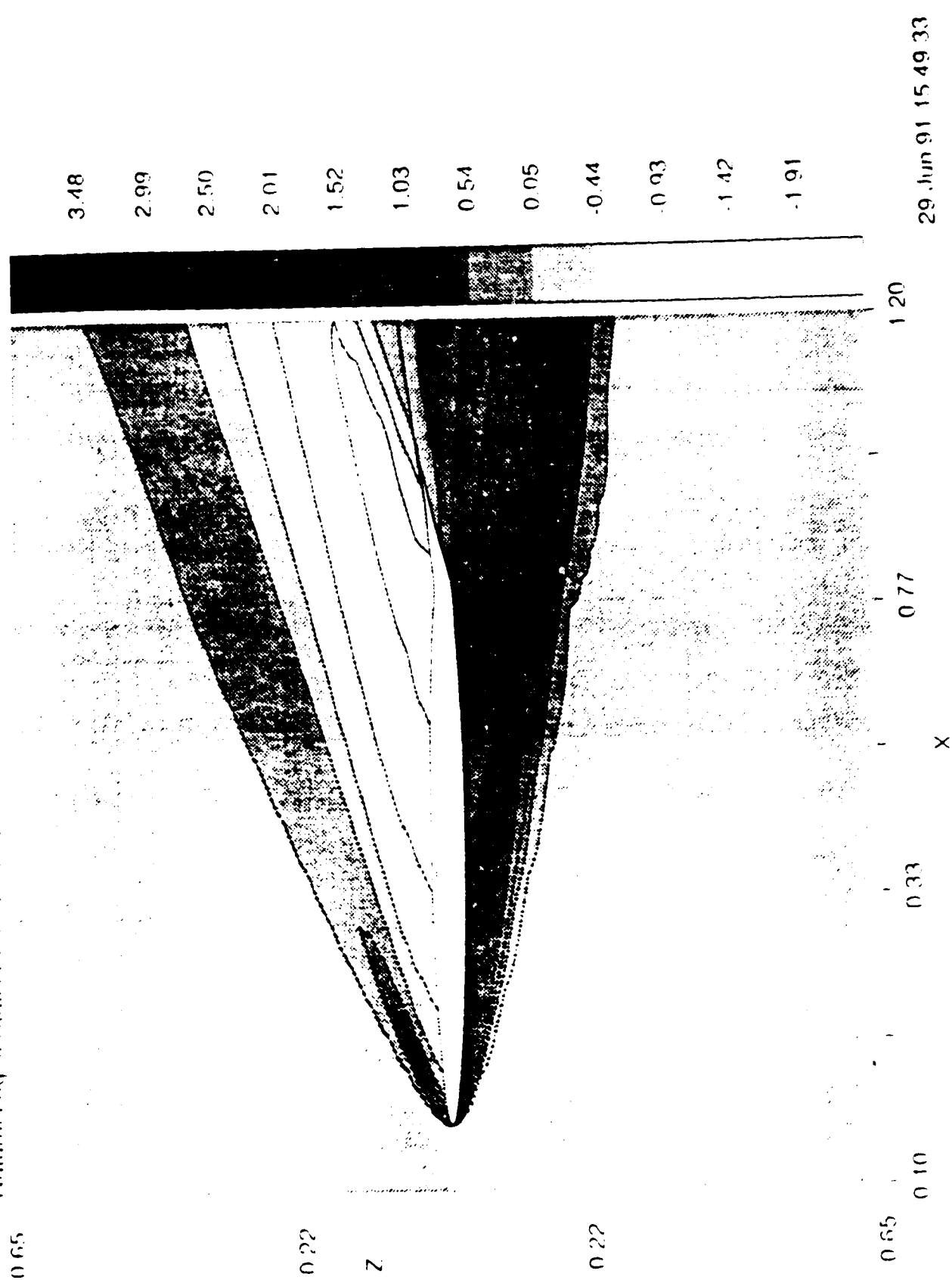
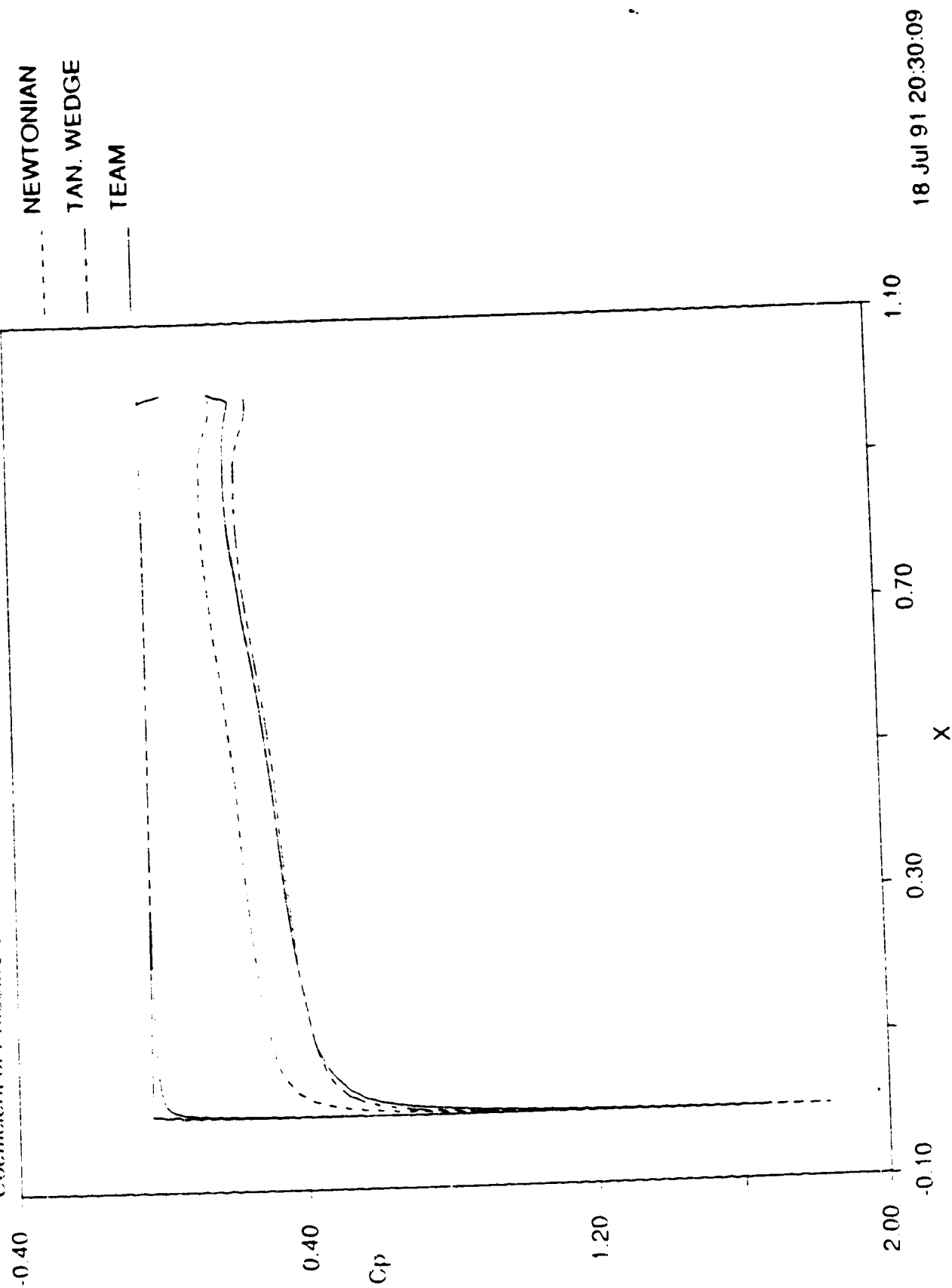


Figure 7b.

29 Jun 91 15 49 33

PSU, Hypersonic Results: $M = 6.83$, $\Lambda = 20$, $D = 0$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



18 Jul 91 20:30:09

Figure 8a.

PSU, IFAM flow field results: $M = 6.83$, $\Lambda = 120$, $D = 10$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

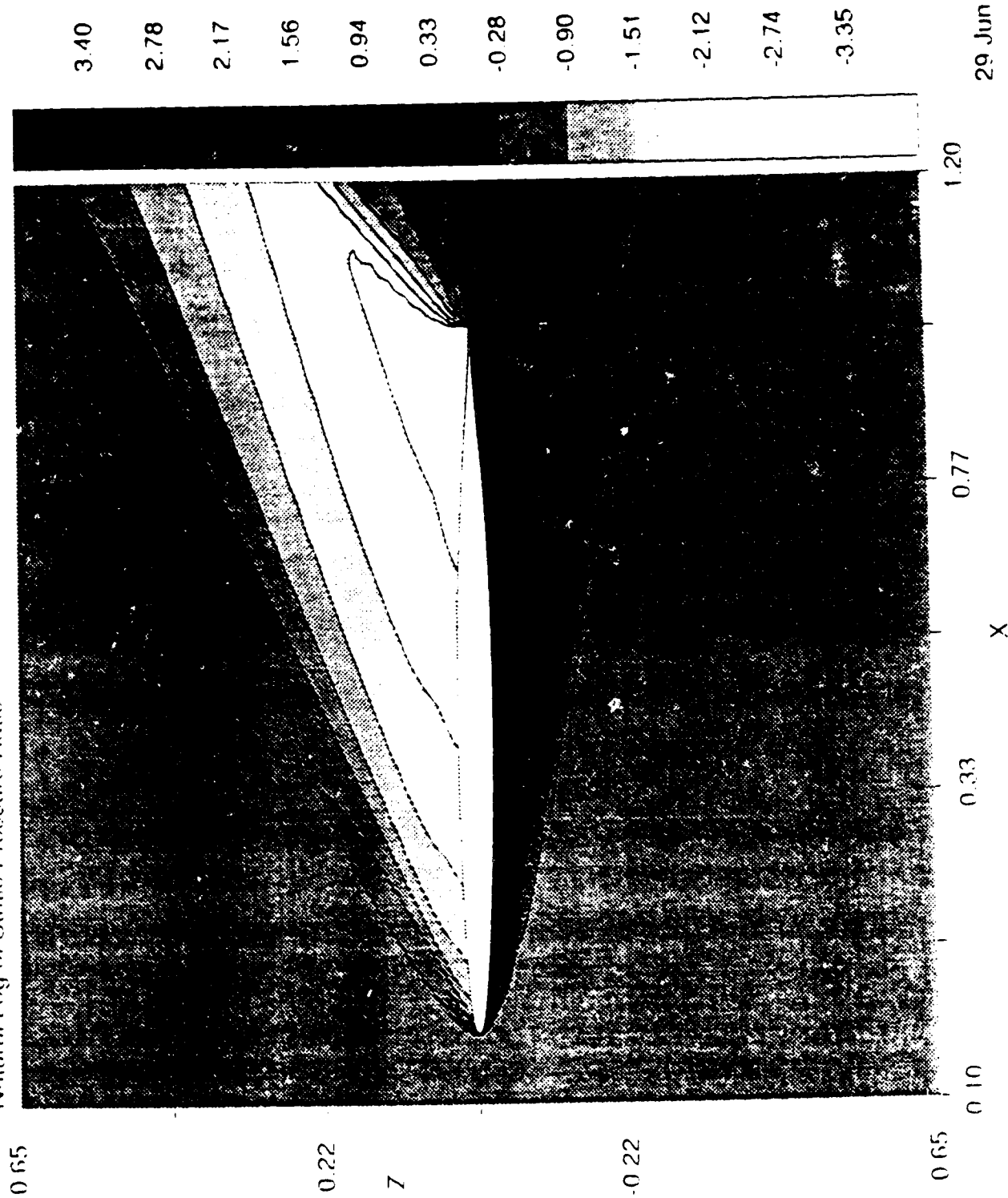


Figure 8b.

PSU, Hyperaero Results: $M = 6.83$, $A = 20$, $D = 10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface

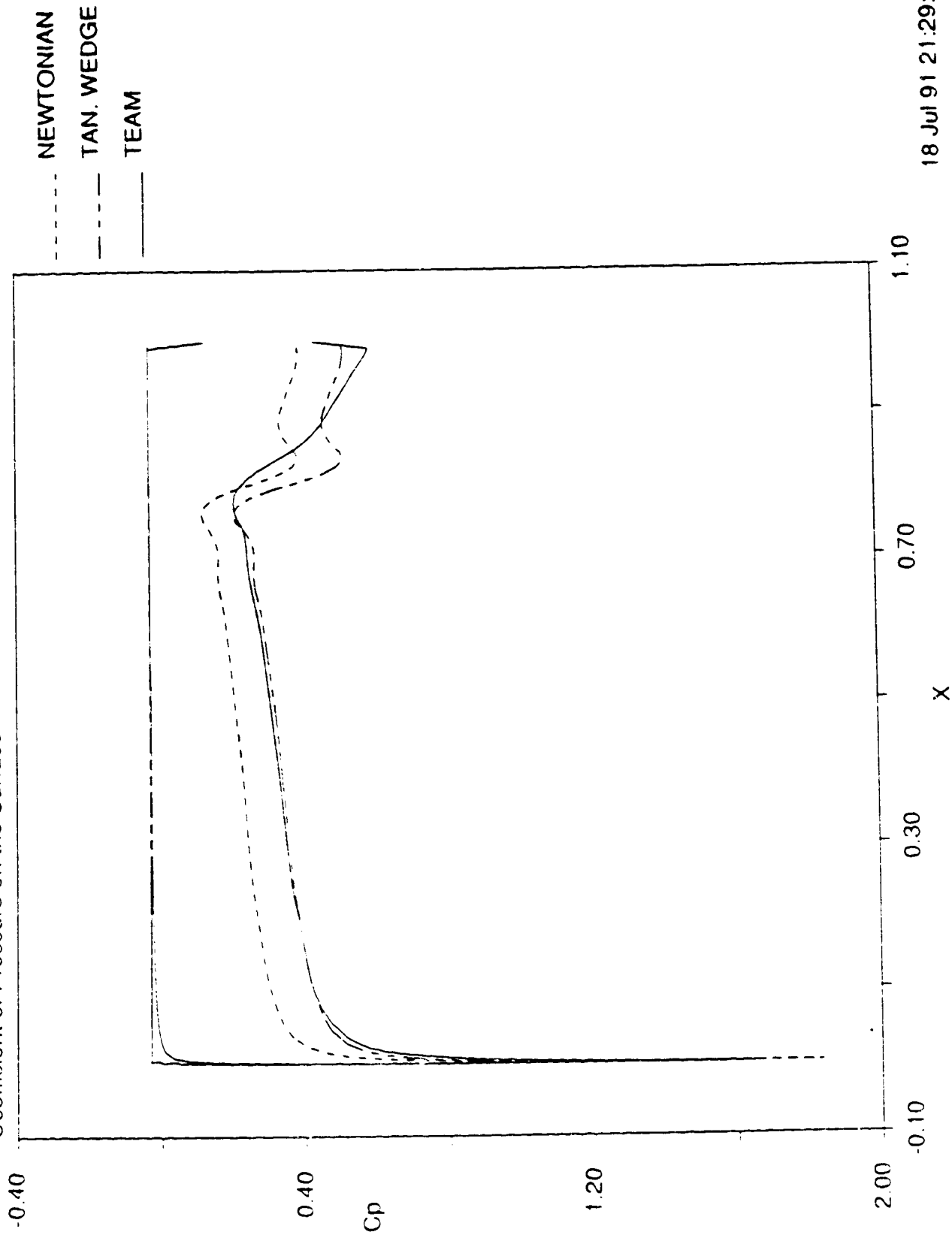


Figure 9a.

18 Jul 91 21:29:42

PSU, TEAM flow field results: $M = 6.83$, $A = +20$, $D = +10$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

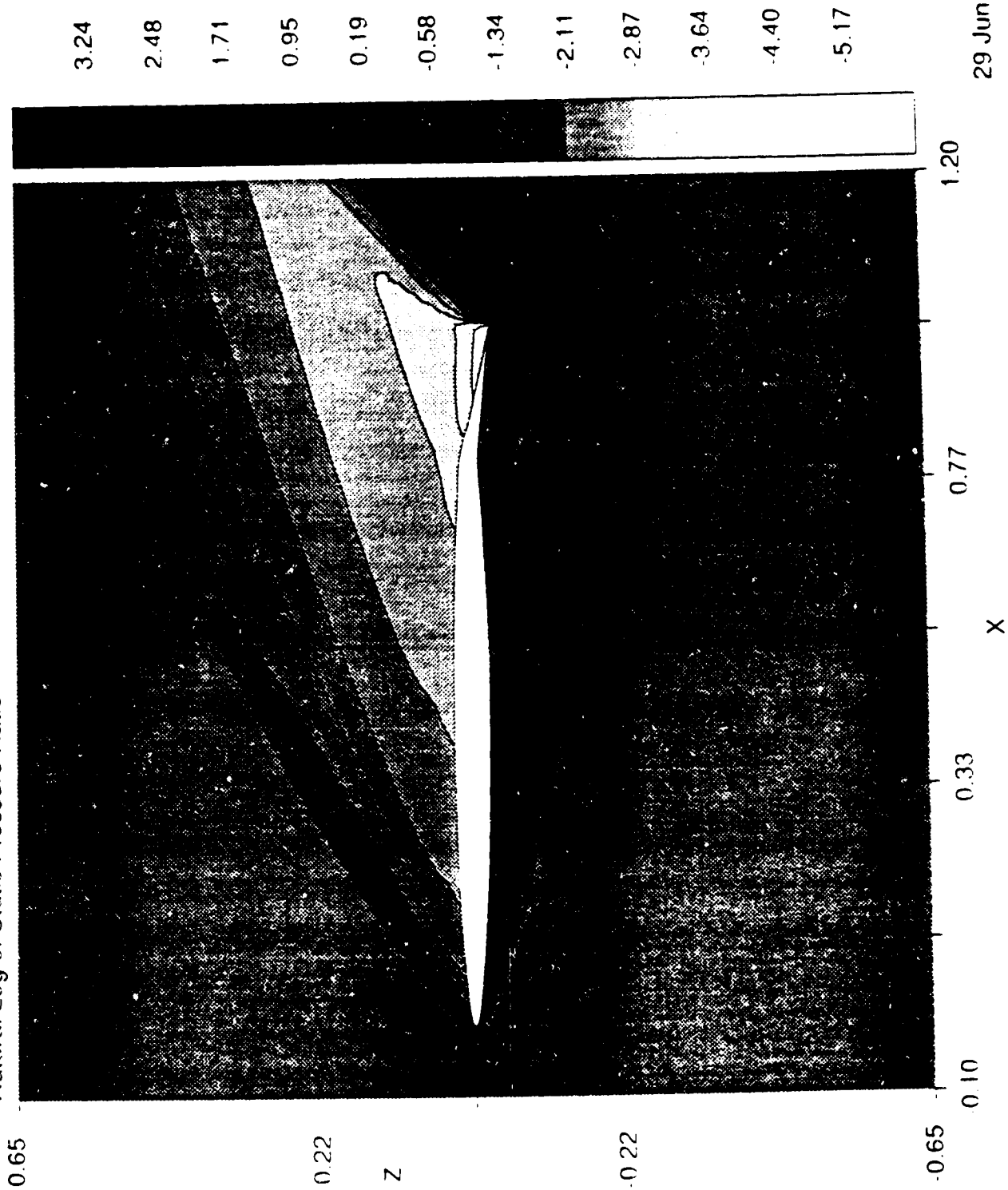
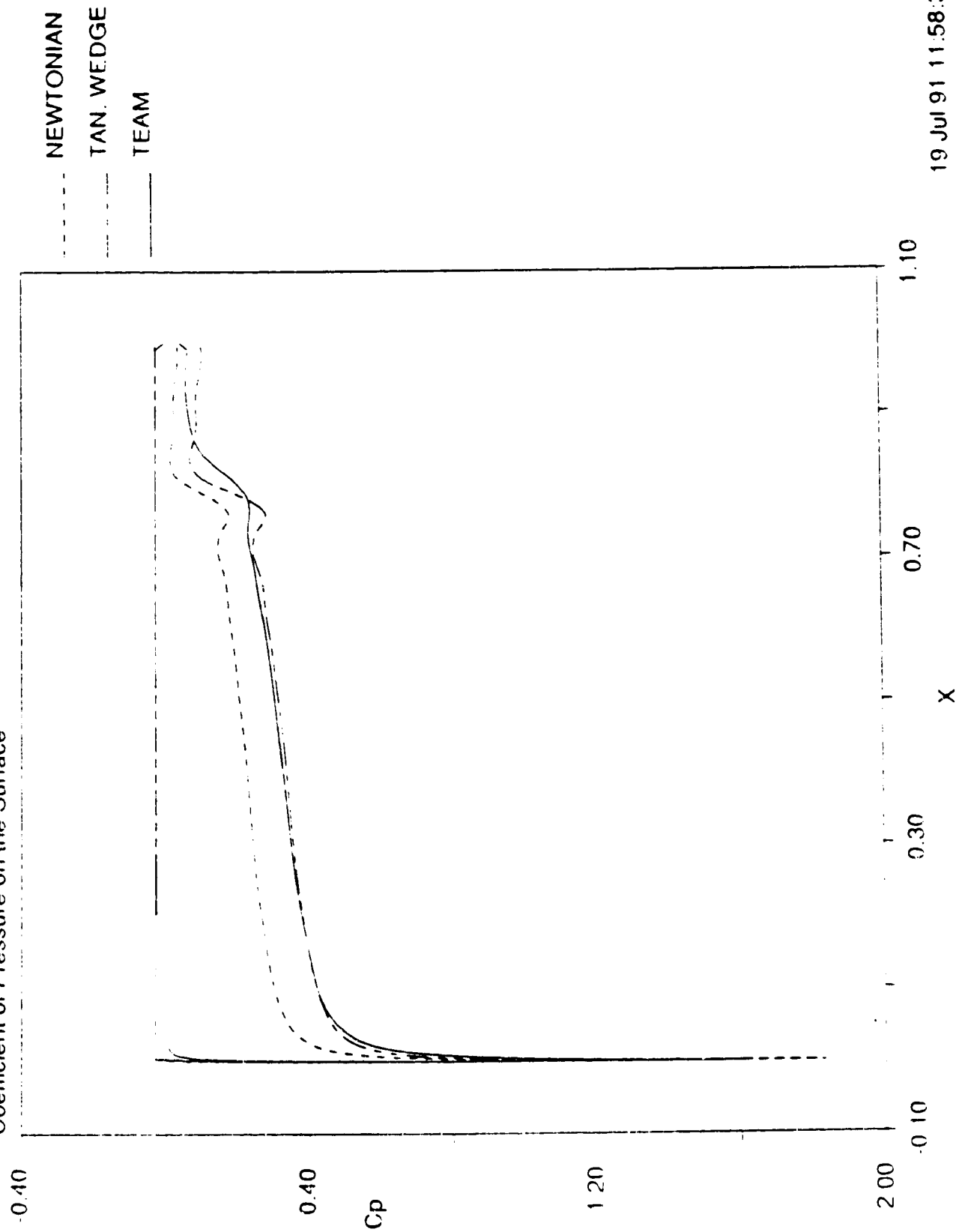


Figure 9b.

29 Jun 91 16:07:41

PSU, Hyperaero Results: $M = 6.83$, $A = 20$, $D = -10$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



19 Jul 91 11:58:33

Figure 10a.

PSU IFAM flow field results: $M = 6.83$, $A = 120$, $D = 10$, Inviscid, Perf Gas
 Natural Log of Static Pressure Ratio

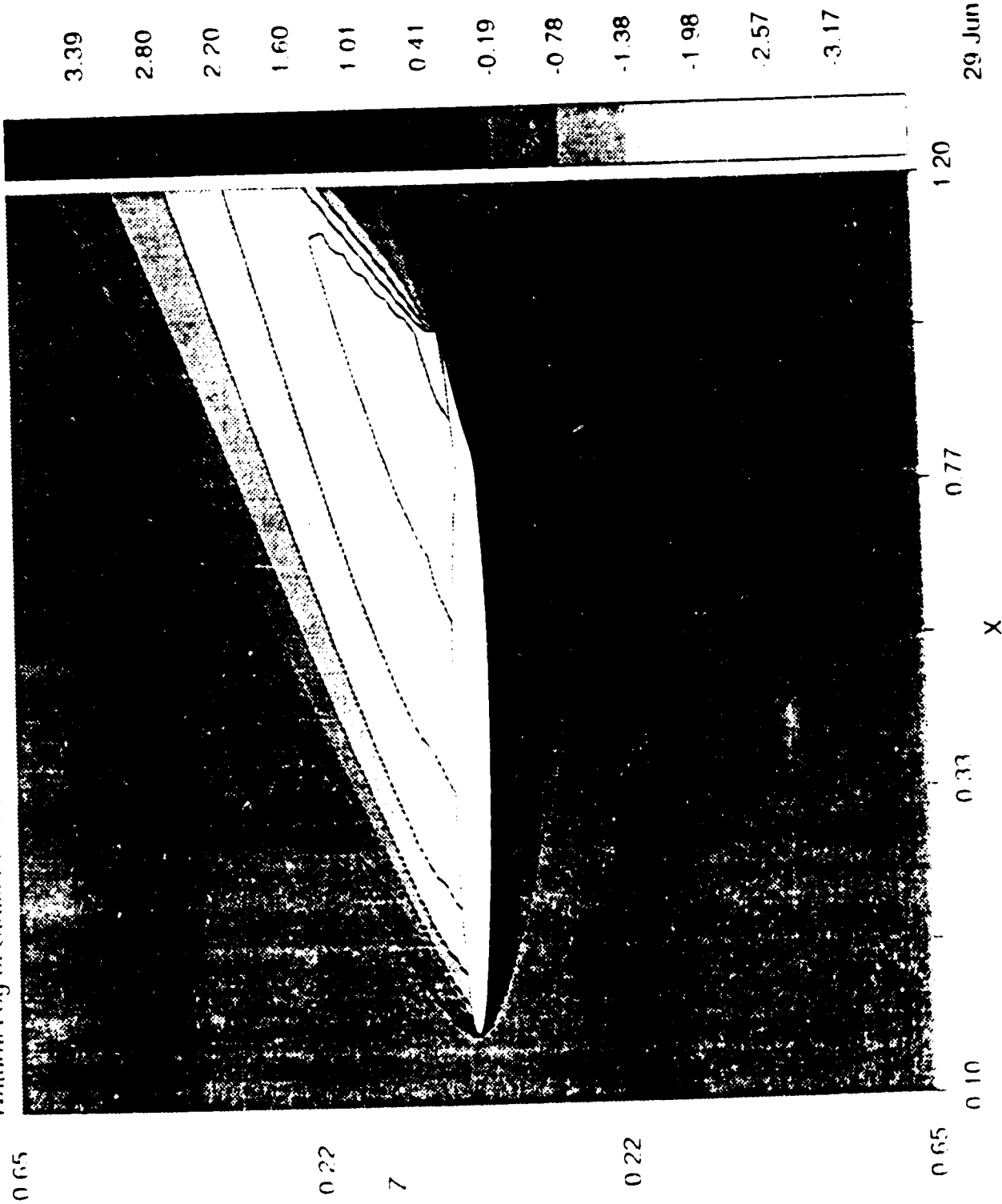
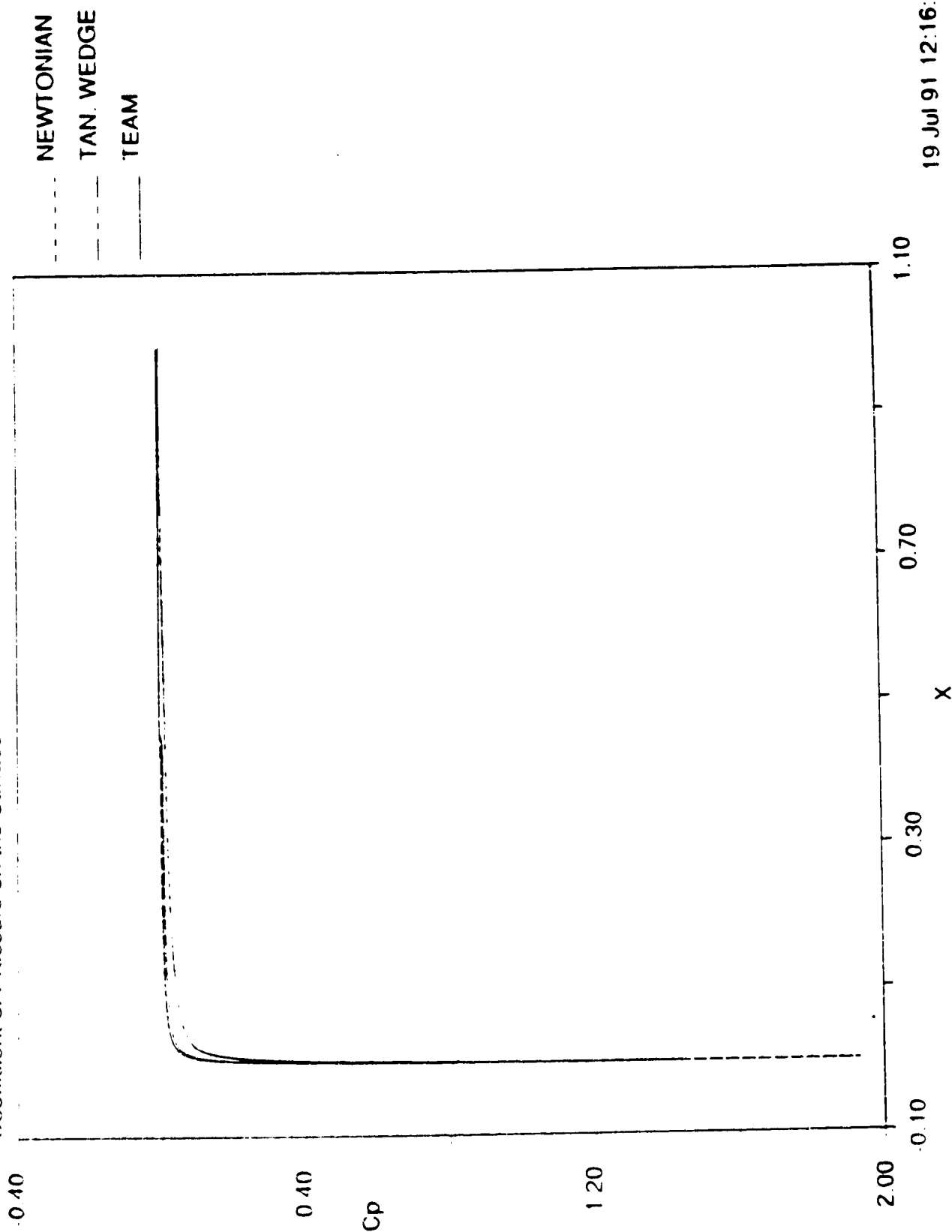


Figure 10b.

29 Jun 91 16:15:07

PSU, Hyperaero Results: $M = 23$, $A = 0$, $D = 0$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



19 Jul 91 12:16:13

Figure 11a.

PSU, IFAM flow field results: $M = 23.00$, $A = +0$, $D = +0$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

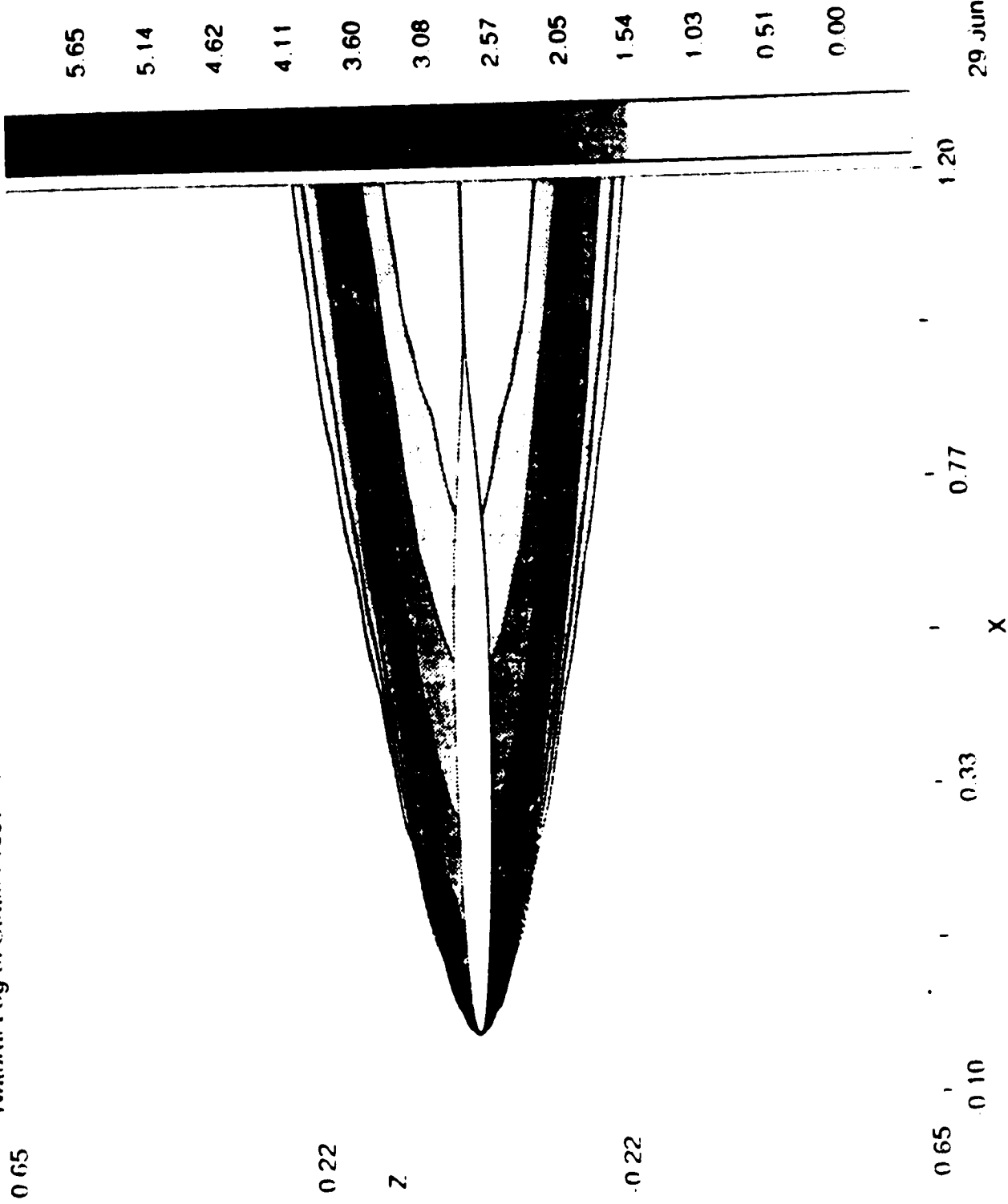
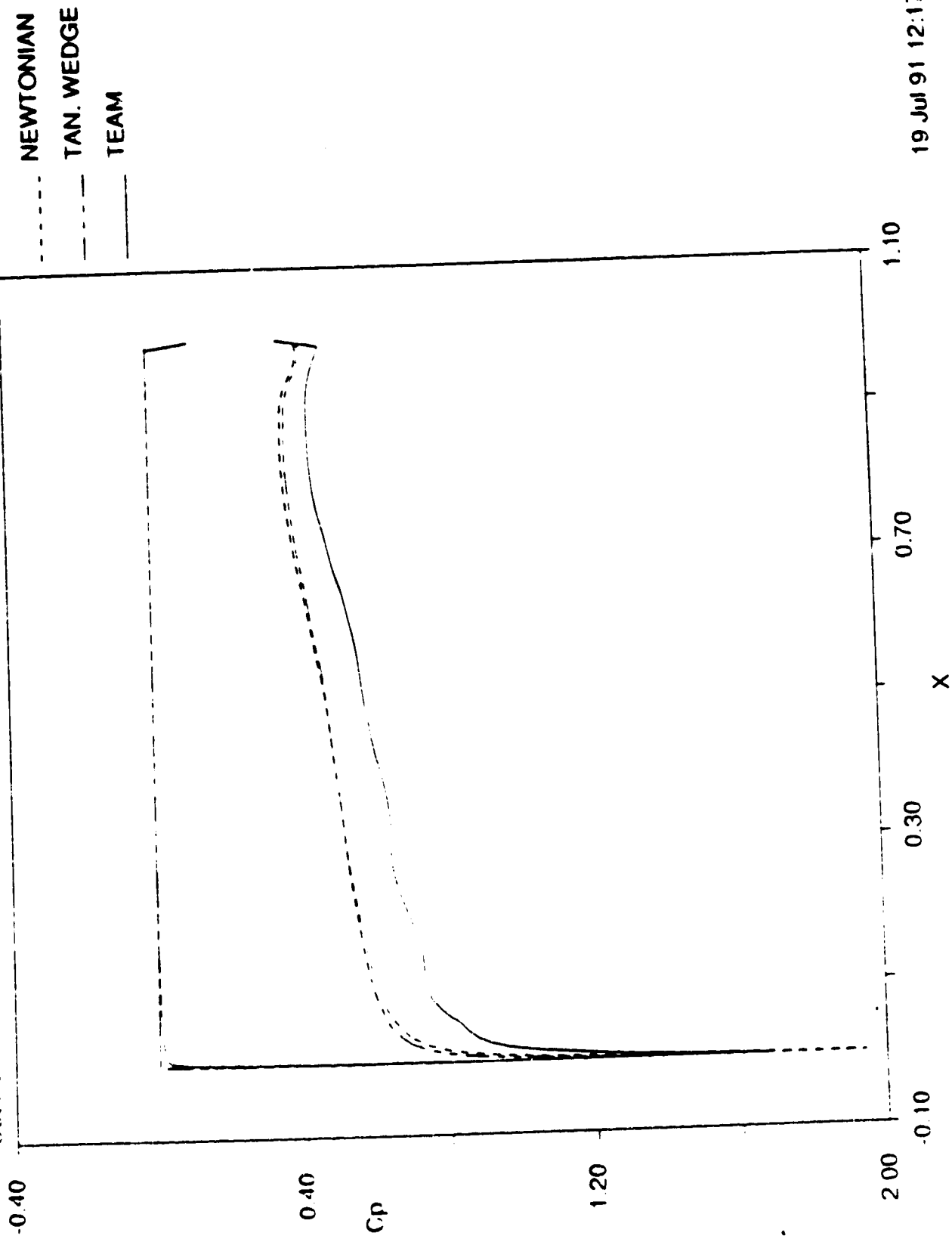


Figure 11b.

PSU, Hyperaero Results: $M = 23$, $A = 30$, $D = 0$ Inviscid, Perf. Gas
Coefficient of Pressure on the Surface



19 Jul 91 12:17:49

Figure 12a.

PSU, TEAM flow field results: $M = 23.00$, $A = +30$, $D = +0$, Inviscid, Perf. Gas
 Natural Log of Static Pressure Ratio

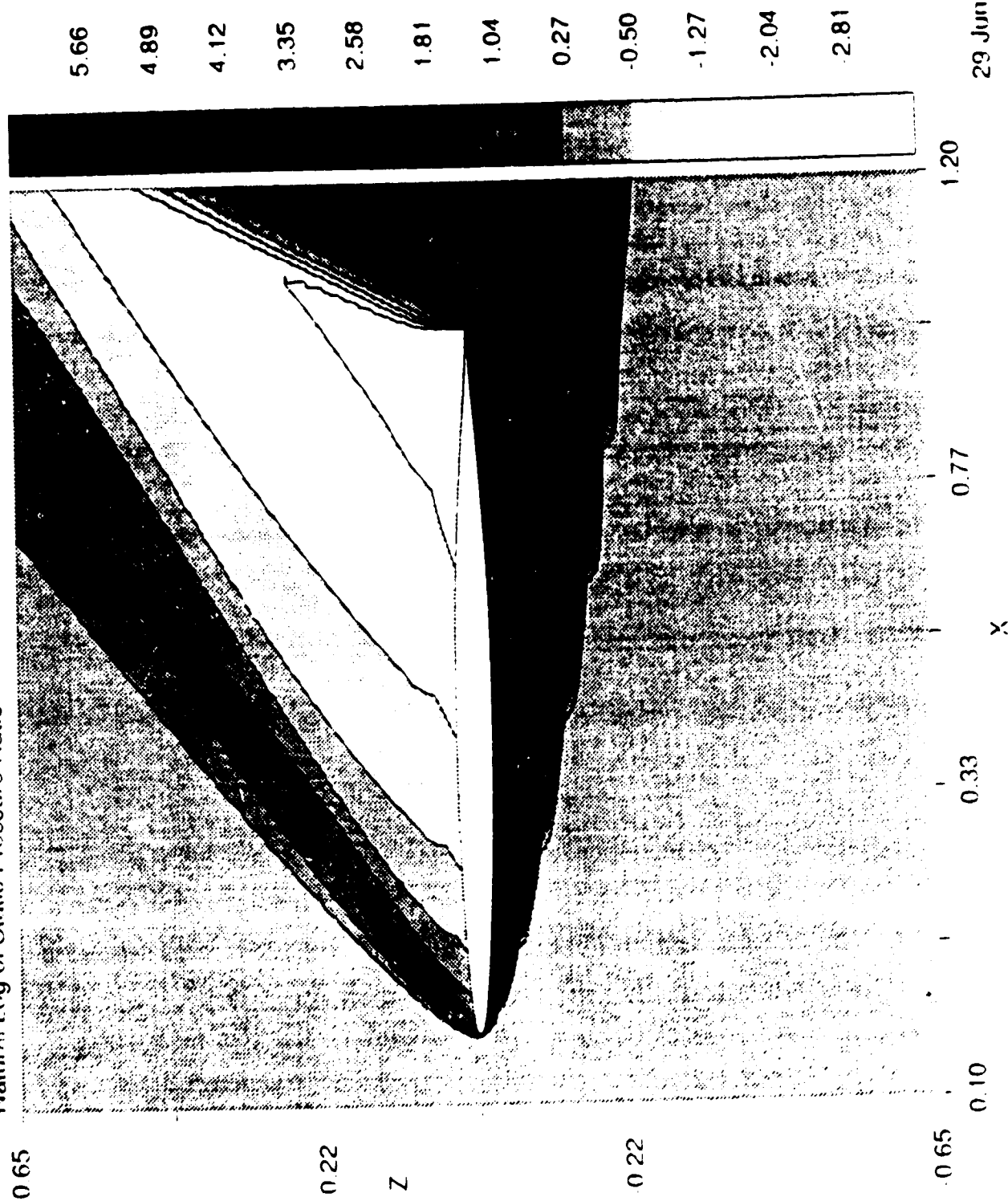


Figure 12b.

29 Jun 91 14:13:35